Advanced Computer Tools for Roadway Hydraulic and Hydrologic Design

Utah Department of Transportation (UDOT #00.05)
And the
Mountain Plains Consortium

by

William J. Grenney
Chandrasekhar Swaminathan
Newell Crookston

Utah State University
Logan, Utah

January 2000

Disclaimer

The contents of this report reflect the views of the authors, who are responsible for the facts and accuracy of the information presented herein. This document is disseminated under the sponsorship of the Department of Transportation, University Transportation Centers Program, in the interest of information exchange. The U.S. government assumes no liability for the contents or use thereof.

Table of Contents

INTRODUCTION 1

PROJECT OBJECTIVES 1

SPECIFIC OBJECTIVES AND TASKS 2

Objective 1 2

Objective 2 2

REVISED OBJECTIVES AND TASKS 2

Objective 3 3

CULVERT HYDRAULICS PROTOTYPE MODEL 4

UDOT WORKSHOP 4

REGIONAL VIDEO CONFERENCE 4

SURVEY OF GIS TYPE HYDROLOGY MODELS 6

GENERAL SURVEY 6

SMS (Surface Water Modeling System) 6

RMS (River Modeling System) 6

GEOPAK DRAINAGE 7

STORMWORKS 7

WMS (Watershed Modeling Program) 7

DISCUSSION 8

RECOMMENDATIONS 11

APPENDIX A: Workbook for the TEL8 Conference A-1

APPENDIX B: Prototype Culvert Module B-1

APPENDIX C: Models Removed During Initial Screening C-1

APPENDIX D: Comparison of GEOPAK and Storm Works D-1

APPENDIX E (available upon request)

INTRODUCTION

Localized flooding and road damage can occur if drainage systems are not properly designed. The Utah Department of Transportation (UDOT) had the HYDRAIN computer library standardized on the U.S. Federal Highway Administration (FHWA) for conducting drainage calculations. However, this suite of computer programs is DOS-based and can be difficult to use effectively by consultants and UDOT engineers. In addition, HYDRAIN does not provide a conventional geographical information system (GIS) module for estimating watershed parameters. The HYDRAIN software currently available does not interface directly with the MicroStation Computer Aided Design (CAD) system being used throughout UDOT. The Hydraulics Division believes that productivity and accuracy could be increased by improving the culvert module and by identifying an improved hydraulic design software program.

Therefore, this project was initiated to develop and select advanced computer programs to improve roadway hydraulic and hydrologic design tools available for use by UDOT engineers and consultants. The development and selection of such computer software that uses engineering analysis and interactive graphics is expected to reduce design and production times and improve quality control.

PROJECT OBJECTIVES

Project objectives were developed under the guidance of the UDOT Technical Advisory Committee (TAC). Members of the TAC for the project included:

Name	Organization	Telephone
Bryan Adams	UDOT, Central Hydraulics	(801) 965-4231
Doug Anderson	UDOT Research and Development	(801) 965-4377
Steve Bartlett	UDOT Research and Development	(801) 965-4377
Abby Fallahi (UDOT Lead)	UDOT, Central Hydraulics	(801) 965-4693
Bill Gedris	FHWA, Utah Division	(801) 963-0182, Ext. 243
Skip Hudson	MK Centennial	(801) 268-9805
Gerald Robinson	UDOT, Region 3 Hydraulics	(801) 227-8062
Denis Stuhff	UDOT Central Hydraulics	(801) 965-4231

The original scope of work for the project was established in July 1998. The general objectives of the project follow:

1) Produce a prototype module for the analysis of culvert hydraulics, including enhanced functionality requested by UDOT.

2) Survey the functionality of the ArcView GIS software for application to hydrologic estimates.

SPECIFIC OBJECTIVES AND TASKS

Objective 1:

Produce a prototype module for the analysis of culvert hydraulics and energy dissipaters.

Task 1: Develop a prototype computer module for culvert hydraulics that incorporates the specific functionality requested by UDOT hydraulic engineers, including adjusted invert, inlet control, outlet control, critical flow, normal flow, full barrel flow, water surface profiles, and energy dissipaters.

Task 2: Provide a user's manual and training for computer module.

Task 3: Provide beta testing and focus group critique of the software products.

Objective 2:

Survey the functionality of the ArcView GIS software for application to hydraulic design.

Task 1: Conduct a survey of features of ArcView as they relate to the use of digital terrain maps (DTMs) for estimating runoff and stream flow conditions for culvert design.

Task 2: Present a summary of features that would be most useful for UDOT design engineers.

REVISED OBJECTIVES AND TASKS

Work progressed on the tasks, and presentations were made at two TAC meetings during summer and fall 1998. A workshop was held at a University of Utah computer laboratory in October 1998 for a group of UDOT hydraulic engineers to critique the prototype model.

At a TAC committee meeting on Jan. 22, 1999, two presentations were made:

1) Features of ArcView for potential use by UDOT for hydraulic design.
2) Revisions to the prototype hydraulic model based on recommendations from the fall workshop.

The TAC engaged in prolonged discussions on the use of ArcView and other GIS products, and decided that it would be impractical for each UDOT hydraulic design engineer to become sufficiently skilled in the use of ArcView to apply it to their day-to-day activities. The committee members believed that GIS software specifically tailored for hydraulic design was available commercially. The TAC committee recommended a shift of the remaining project resources as follows:

1) Discontinue work on the prototype hydraulic model (Objective 1).
2) Discontinue work on the assessment of ArcView (Objective 2).
3) Conduct a survey of the commercial GIS software products available specifically for hydraulic design (New objective).

The new objective was further defined as follows:

Objective 3:

Survey hydraulic design software products that have the potential to interface with UDOT roadway design procedures and standards and recommend one for use by UDOT.

Task 1: Conduct a survey of hydraulic software products that currently are available and have the potential to support UDOT roadway design procedures and standards.

Task 2: Present a summary of features of selected software products and recommend use of one for UDOT.

CULVERT HYDRAULICS PROTOTYPE MODEL

UDOT WORKSHOP

The fall 1998 workshop, held in the Engineering Computer Laboratory at the University of Utah, was conducted for the Utah Department of Transportation and covered the following topics:

Software features of the culvert model

Prediction of scour at a structure outlet

Design of internal and external energy dissipaters

Hydraulic jump

REGIONAL VIDEO CONFERENCE

When Project Objective No. 1, Development of a Culvert Module, was about 50 percent complete, a teleconference was presented by Dr. Grenney to demonstrate the new computer modules, as well as to get focus group critique and feedback from practicing design engineers on how the modules could be improved. The Utah, Wyoming, and N.D. Departments of Transportation participated in the eight-hour video conference. Topics included:

Summary of the new modeling processes being implemented.

Review of the fundamentals of flow through culverts and demonstration of model features for:

Free surface flow

Hydraulic jump

Water surface profiles

Demonstration of features of the new Culvert Model, including:

Roadway Properties Window

Culvert Properties Window

Flow Properties Window

Results Window

Demonstration of new features for filing, printing, and data sharing.

Demonstration of features for the Energy Dissipater Module, including:

Scour hole

Internal dissipaters

External dissipaters

Hydraulic jump

The workbook for the teleconference, which includes a discussion of the fundamental processes upon which the Culvert Model is based, is provided as Appendix A

The Prototype Culvert Module is a tool for calculating information about flows in open channels and closed barrels commonly used for culverts, including:

The shape geometry for sixteen standard shapes

The hydraulic properties of a user-defined shape as well as the standard shapes

The water surface profile for a specified upstream or downstream boundary condition.

The User's Manual for the Prototype Culvert Module is presented in Appendix B

The source code for the computer module is presented in the last Appendix of this report (Appendix E).

SURVEY OF GIS TYPE HYDROLOGY MODELS

GENERAL SURVEY

A survey of potential software products was made using the Internet. A summary of software products, including short descriptions of applications and features that were screened out from further investigated is presented in Appendix C.

Software products that were evaluated in depth included: the Surface Water modeling System (SWMS), RMS (River Modeling System), GEOPAK Drainage, StormWorks, and WMS (Watershed Modeling Program).

SMS (Surface Water Modeling System)

This program was recommended by the FHWA as a two-dimensional modeling program. It was developed by the Engineering Computer Graphics Laboratory at Brigham Young University (BYU) (HYPERLINK http://ecgl.byu.edu/index.html http://ecgl.byu.edu/index.html) and is marketed and supported by BOSS International (http://www.bossintl.com). The software models the water surface elevation, flow velocity, contaminant transport and dispersion, and sediment transport and deposition for complex two-dimensional horizontal flow problems.

RMS (River Modeling System)

This program, marketed by BOSS International, is used to compute water surface profiles for modeling bridges, culverts, spillways, levees, bridge scour, floodway delineation and reclamation, stream diversions, split flows, and channel improvements using the U.S. Army Corps of Engineers HEC-2 and HEC-RAS water surface profile models (see Appendix D) in AutoCAD (http://www.bossintl.com).

GEOPAK DRAINAGE

This program module, which is part of a civil engineering suite of products, is used to design, analyze, and visualize storm water flow, with drainage features integrated with road and site design tools. The program can handle multiple drainage networks, comprising any number of topologically-connected areas, inlets, pipes, and ditches. It uses the MicroStation graphics environment for the definition, design, and review of drainage systems (http://bentley.com/products/ceproducts/).

STORMWORKS

Intergraph's StormWorks, which integrates with a suite of other Intergraph civil engineering applications, is a comprehensive program for surface water collection, transport, and disposal. The program utilizes 3-D modeling and interactive graphics (http://www.intergraph.com/iss/products/civil).

WMS (WATERSHED MODELING PROGRAM*)*

The WMS software provides a comprehensive environment for hydrologic analysis of watershed systems. Developed in cooperation with the U.S Army Corps of Engineers Waterways Experiment Station and marketed and supported by BOSS International (http://www.bossintl.com), WMS provides graphical tools for use in delineation of watersheds and flood plains. The U.S Army Corps of Engineers HEC-1 and the U.S. Soil Conservation Service TR-20 hydrologic routing programs (see Appendix D) may be set up and viewed in a user-friendly graphical environment. Interfaces to the USGS National Flood Frequency program and the Rational Method Equation provide other modeling options (http://ripple.wes.army.mil/software/).

DISCUSSION

Results of the evaluation of the software packages were presented to the TAC at a meeting at Utah State University. At this meeting, two programs, StormWorks and GEOPAK Drainage, were selected for an in-depth review and evaluation. Complete features of the software are presented in Appendix D, Chapter 1 for GEOPAK Drainage and in Appendix D, Chapter 2 for StormWorks.

Procedures that were established to evaluate the software products included:

Identification of the purpose of the software

Demonstration of the software by the software developer

Testing of the software using a known hydrologic design

Evaluation of the characteristics and features of the software, including:

Ease of learning

Functionality

Applicability to the task

Availability of technical support

A summary of the features, approaches, and functionality was developed to illustrate differences between StormWorks and GEOPAK Drainage. Various features and approaches presented by these models were compared and contrasted.

Platform:

GEOPAK Drainage is invoked from a 2-D MicroStation Design file, whereas StormWorks can work on two-dimensional and three-dimensional design files. Three-dimensional design files offer greater leverage for designing a drainage model.

Surface/Geometric data:

The Surface/Geometric data define the spatial extent and topological features of a drainage basin and, as such, is the first step in a design process. GEOPAK Drainage uses TIN

files to store terrain data, while StormWorks stores surface data as either a DAT file, a DTM file or a TTN file.

In the design of a drainage network, roadway alignments, vertical profiles, and Digital Terrain Models (DTMs) play an important part. GEOPAK Drainage is closely integrated with other GEOPAK civil engineering design software such as GEOPAK Bridge (modeling and design), GEOPAK Survey (survey data handling), and GeoTrain (digital terrain modeling). In similar fashion, terrain files are interchangeable between StormWorks, InRoads (transportation design), SiteWorks (site design), InWater (water distribution), InSewer (wastewater collection) and other Intergraph products. Therefore the use of StormWorks may be preferable if terrain models have already been developed using other Intergraph software such as InRoads. Surface files are saved with the extension .dtm and geometry files are saved with the extension .alg in InRoads. These are the same files used to define the basin in StormWorks.

Hydrologic features:

Rainfall Data: For rainfall data, StormWorks uses intensity-duration frequency tables or rainfall-time of concentration or intensity equations for obtaining rainfall data. GEOPAK Drainage also uses intensity-duration frequency tables. In addition, GEOPAK Drainage has a wide range of equations from which to calculate rainfall intensity, including:

i= f{a,b, c,Tc}

i= f{a,b,c,frequency,Tc}

i= f{a,b,c,ln[tc]}

Peak Runoff: GEOPAK Drainage uses the rational method for peak runoff computations and frequency dependent peak factors for runoff coefficients. StormWorks uses the modified rational formula or the SCS methodology.

Land Characteristics: Land uses within a project can be assigned in GEOPAK Drainage. Thus land can be categorized as grass, commercial, pavement, building, industrial, etc. As drainage areas are delineated, software determines the proportion of land uses, computes

subsequent runoff coefficients, and automatically assigns the hydrologic parameters for the drainage areas. The approach used in StormWorks is slightly different. In StormWorks, the type of land cover—asphalt, concrete, sandy lawn, clay, roofs, etc., can be selected along with their corresponding runoff coefficients. The former approach is quite generalized, whereas the latter approach is more down-to-earth and well-defined for most situations.

Drainage Area Parameters: The computational method used for drainage area by StormWorks is either the modified rational formula or the SCS Unit hydrograph method. GEOPAK Drainage uses the rational method.

Time of Concentration: The time of concentration, Tc, can be determined using either the Kirpich or FAA methodologies in StormWorks. In GEOPAK Drainage, the value of Tc is input directly.

Hydraulic Features:

Channels: In StormWorks, the channel can be designed by shape, trapezoidal, V-shaped or rectangular. Additionally, the channel material such as concrete, earth, or wood can be specified. In case of GEOPAK Drainage, channels are designed either by fixed geometry or based on cross-section. Therefore, trapezoidal or irregular ditches can be designed, but the material cannot be specified. Slope and velocity constraints can be defined in both models. In addition, GEOPAK Drainage can set constraints on the channel rise.

Pipes: Both models provide options for the type of pipe and the material used. In the case of StormWorks, the software automatically displays error flags if design limitations are exceeded. In both programs, pipe and ditch profiles can be created showing hydraulic and energy gradelines.

Flow equations: GEOPAK Drainage uses the Manning flow equation while StormWorks offers a choice between the Manning, Darcy-Weisbach and Colebrook-White equations. It should be noted that in StormWorks, pipes and culverts are designed using the Darcy-Colebrook equation, while open channels and gutters are always designed using the Manning equation.

Pumps: Pump parameters can be specified and included in the drainage design in the case of StormWorks while GEOPAK Drainage does not have such capability.

Management Capabilities:

Both GEOPAK Drainage and StormWorks offer extensive reporting facilities. The reporting capabilities of GEOPAK Drainage are superior compared to StormWorks. StormWorks offers queried reports whereas customized reports can be generated using GEOPAK Drainage. This format can be saved for reuse. StormWorks has a management tool that can be used to monitor the installation and service dates of selected structures. The reporting capabilities of StormWorks can be enhanced if it is used in conjunction with DraftWorks, an Intergraph drawing utilities package provided in its civil engineering software package.

RECOMMENDATIONS

A workshop was held at Utah State University to present results of the Project Objective No. 3, Survey and Recommendation of Hydraulic Design Software. A demonstration of the features of the two software packages selected for final review (GEOPAK Drainage and StormWorks) was presented.

Pros and Cons of the software were analyzed. It is worth noting here that both StormWorks and GEOPAK Drainage offer extensive capability and, as such, neither are inferior to the other. An important factor discussed was that UDOT uses Inroads, an Intergraph product for designing their roadways. It was thought that the digital terrain models developed using Inroads could be directly input into StormWorks precluding the need to recreate terrain maps. This also would entail uniformity among the various departments in UDOT.

After discussion and questions, StormWorks was selected as the preferred software program for use by UDOT primarily because of its connectivity with the other design packages offered by Intergraph.

APPENDIX A

TEL 8 TELECONFERENCE ON

Culvert and Energy Dissipator

Design Computer Modules

TEL 8 TELECONFERENCE

Preview of the next release of the FHWA HY8 Culvert and Energy Dissipator Design Computer Models

Background.
Dr. Grenney is currently leading a project to reprogram the FHWA HY8 Culvert and Energy Dissipator Design Computer Modules. The project is about 80% complete. Dr. Grenney is offering this conference in order to demonstrate the new computer modules, as well as to get feedback from practicing design engineers on how the modules can be improved before final release.

Topics

1) Summarize the new modeling process being implemented.

2) Review the fundamentals of flow through culverts and demonstrate the model features for:
   - Free surface flow
   - Hydraulic jump
   - Water surface profiles

3) Demonstrate features of the new Culvert Module
   - Roadway Properties Window
   - Culvert Properties Window
   - Flow Properties Window
   - Results Window

4) Demonstrate new features for Filing, Printing and data sharing.

5) Demonstrate the features for the Energy Dissipator Module
   - Scour hole
   - Internal dissipators
   - External dissipators
   - Hydraulic jump

Program

Day 1	
8:00 - 9:00	Teleconference Testing and setup
9:00 - 10:30	Presentations on Topics 1 and 2
10:30 -10:45	Break
10:45 -12:00	Independent work at each site. Dr. Grenney will be available to answer questions
12:00 - 1:00	Lunch
1:00 - 3:00	Presentations on Topics 3 and 4
3:00 - 3:15	Break
3:15 - 4:30	Independent work at each site. Dr. Grenney will be available to answer questions
Day 2	
8:00 - 8:30	Teleconference Testing and setup
8:30 - 10:30	Open discussion and presentations on Topic 5
10:30 -10:45	Break
10:45 -12:00	Independent work at each site. Dr. Grenney will be available to answer questions
12:00 - 1:00	Lunch
1:00 - 3:00	Open discussion, bug reports, suggestions for improvements.
	Adjourn

Site Requirements

Each site must provide at least one PC running Window 95 for use by the students in the vicinity of the teleconferencing room.

Conference Materials

Software will be sent to each site several days before the conference. The software should be installed and running prior to the start of the conference.

Each site will provide their own copies of the following FHWA manuals for the conference participants to refer to:

HEC 15, Hydraulic Design of Highway Culverts
HEC 14, Hydraulic Design of Energy Dissipators for Culverts and Channels.

Conference Cost

$100 per site should be sent to the Cynthia Prante at the Utah Transportation Center, Utah State University, Logan, UT 84322-8200 in order to cover operating and material costs.

INTRODUCTION

DEFINITION OF TERMS

List of Symbols

Following is the list of symbols used in this manual.

A: Cross-sectional area of the water in a channel/culvert.

D_r: The rise of the cross section of a channel/culvert.

D_s: The span of the cross section of a channel/culvert.

D_t: The distance from the top of fill in the bottom of the channel/culvert to its top.

D_f: The depth of the fill in the bottom of the channel/culvert.

E: The energy head ($Y + H_p + H_v$).

E_i: The energy head just inside the barrel at the inlet.

E_u: The energy head just upstream from the headworks.

F: Froude number.

g: Acceleration of gravity.

H: Hydraulic head ($Y + H_p$).

H_i: Hydraulic head just inside the inlet.

H_o: Hydraulic head just inside the outlet.

H_p: Pressure head.

H_{pi}: Pressure head just inside the inlet.

H_{po}: Pressure head just inside the outlet.

Hv: Velocity head.

Hvi: Velocity head just inside the inlet.

Hvo: Velocity head just inside the outlet.

h_i: Head loss at the inlet.

K_i: Minor loss coefficient at the inlet.

L: Slope distance from the inlet to some point along the channel/culvert.

L_J: Length of hydraulic jump.

L_o Slope distance from the inlet to the outlet.

n: Manning's roughness coefficient.

n_c: Composite Mannings roughness coefficient

P: Wetted perimeter.

Q: Flow.

Q_{PB}: Flow per barrel in a multi barrel culvert.

R: Hydraulic radius.

S_e: Slope of the energy grade.

S_o: Slope of the bottom of the channel/culvert.

T: Top width of the water surface.

V: Water velocity.

X: Horizontal distance from the inlet to some point along the channel/culvert.

X_o: Horizontal distance from the inlet to the outlet.

Y: Depth of water.

Y_1: Lower conjugate depth.

Y_2: Upper conjugate depth.

Y_{bar}: The distance from the water surface to the centroid of the wetted area.

Y_c: Critical depth.

Y_i: Depth of water just inside the inlet.

Y_{full}: The depth of water when the section is full (equal to D_t)

Y_n: Normal depth.

Y_o: Depth of water just inside the outlet.

Y_{tw}: Tail water depth.

Y_u: Depth of water upstream from the culvert inlet.

z: Run over rise of the slope of the side of a channel/culvert.

ω: Specific (unit) weight of water.

Figure 1 show three typical cross sections. The Ellipse is defined by the rise (D_r) and span (D_s). The Trapezoid is defined by the span (D_s) equal to the bottom width and the run-over-rise for the sides (z). The Rounded Triangle is defined by the span (D_s) at the point where the parabolic

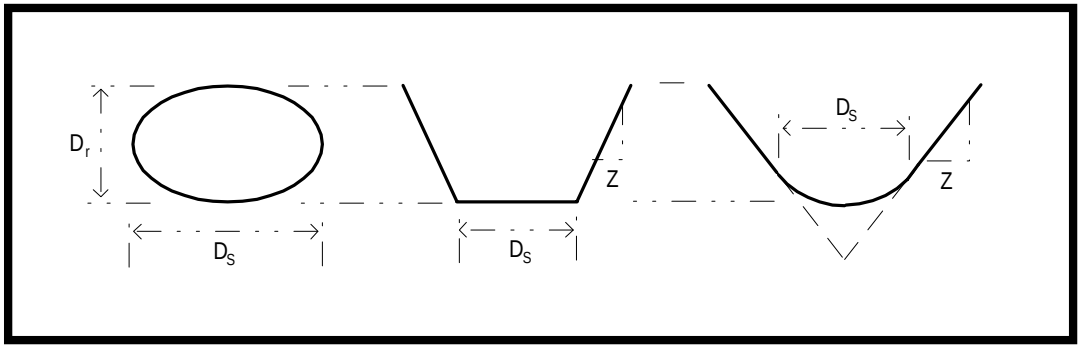


Figure 1. Typical cross sections for channels and culverts.

bottom becomes tangent to the sides of the triangle and the run-over-rise of the sides of the triangle (z). For an open channel, the rise (D_r) is the maximum depth before the flow overtops the sides.

General Depth Relationships

Figure 2 shows the longitudinal profile of a typical culvert on a mild slope. "Dr" is the rise of the culvert section. "D_t" is the distance from the top of the fill material in the bottom of the culvert to the top of the culvert. "Y_u" is the depth of the water just upstream from the inlet. In this example the culvert is on a mild slope and so the normal depth (Y_n) is greater than the critical depth (Y_c). The water surface drops from Y_u to Y_n as it passes through the inlet due to energy loss through the headworks. The flow proceeds downstream at normal depth until it nears the outlet. The tail water (Y_{tw}) is lower than the critical depth, and so the water surface just inside the outlet (Y_o) dips through critical depth.

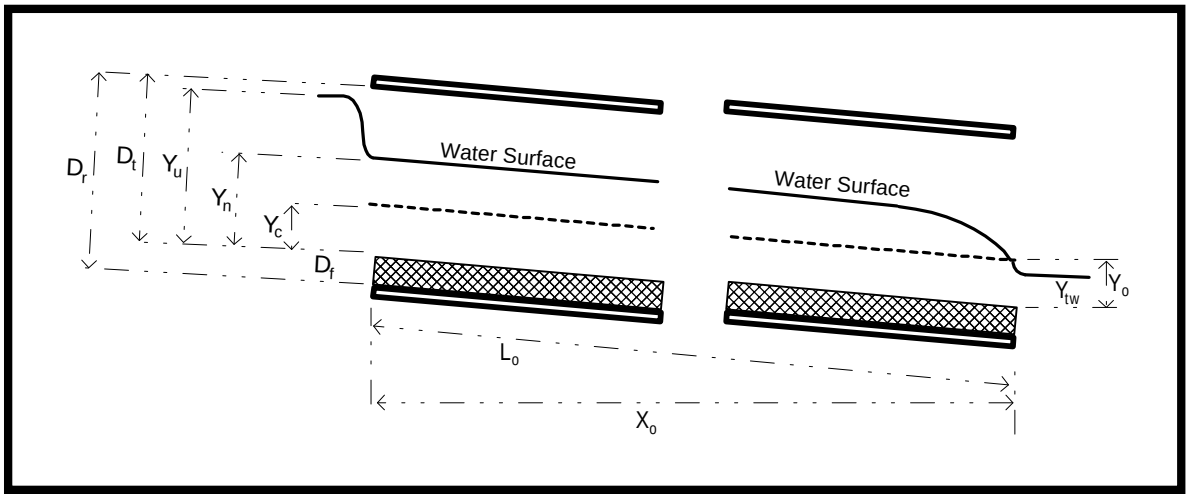


Figure 2. Typical culvert longitudinal profile on a mild slope.

The computer module contains a user option to restrict the depth just inside the outlet (Y_o) from ever becoming less than $(Y_c + D_t)/2$.

Figure 3 shows the longitudinal profile for a culvert on a steep slope. The critical depth (Y_c) is greater than the normal depth (Y_n). The upstream depth (Y_u) dips rapidly to critical depth. From there it dips slowly towards the normal depth as an S2 curve. It is assumed that the depth just inside the inlet (Y_i) is equal to the critical depth.

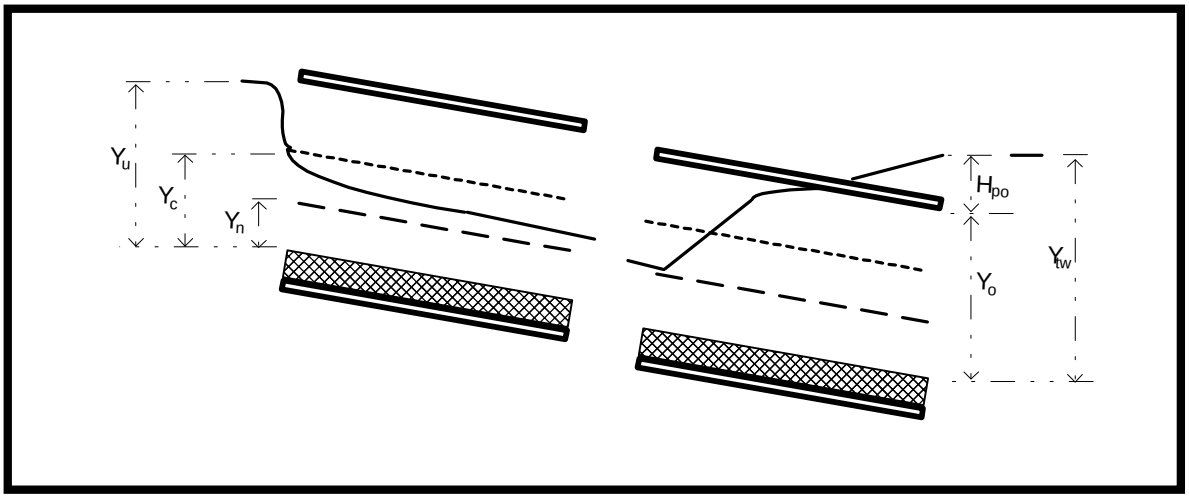


Figure 3. Typical culvert longitudinal profile on a steep slope.

Figure 3 shows the tail water (Y_{tw}) greater than the full flow depth of the culvert (Y_{full} and D_t). It is assumed that the hydraulic head just inside the outlet (H_o) equals the depth of the tail water. Because Y_o is limited to the depth of the barrel, a pressure head (H_{po}) develops just inside the outlet equal to the difference between Y_{tw} and Y_o. Full barrel flow occurs upstream until the pressure head diminishes to zero at which point free surface flow occurs. An S1 backwater curve will extend upstream.

The depth upstream is nearly normal and must cross the critical depth in order to meet the backwater curve. Consequently, a hydraulic jump will occur with lower conjugate depth about

equal to normal depth. The position of the jump will be at the point where the upper conjugate depth meets the backwater curve.

STEADY UNIFORM FLOW

Mannings Formula

$$Q = \frac{1.486}{n_c} R^{2/3} S_e^{1/2}$$

Composit Roughness Coefficient

Option 1:

$$n_c = \left(\frac{\sum_i P_i n_i^{1.5}}{\sum_i P_i} \right)^{2/3}$$

Option 2:

$$n_c = \left(\frac{\sum_i P_i n_i^{2}}{\sum_i P_i} \right)^{1/2}$$

Option 3:

$$n_c = \left(\frac{\sum_i P_i}{\sum_i \frac{P_i}{n_i}} \right)$$

Full Barrel Flow

Mannings equation is used for full barrel (pressure) flow.

Hydraulic Jump

Froude Number.

$$F = \sqrt{\frac{V^2}{gY_h}} = \sqrt{\frac{Q^2T}{gA^3}}$$

Where Y_h is the mean depth defined by:

$$Y_h = \frac{A}{T}$$

Energy Equation (Bernoulli Equation).

$$H_{e1} + Y_1 + H_{p1} + H_{v1} = H_{e2} + Y_2 + H_{p2} + H_{v2} + h$$

$$E_1 = H_{e1} + Y_1 + H_{p1} + H_{v1}$$

$$E_2 = H_{e2} + Y_2 + H_{p2} + H_{v2}$$

Momentum Equation.

$$\omega A_2 \overline{Y}_2 \cos\theta - W \sin\theta - \omega A_1 \overline{Y}_1 \cos\theta = \frac{\omega V_1 Q}{g} - \frac{\omega V_2 Q}{g}$$

$$\frac{W}{\omega} = c \frac{A_1 + A_2}{2} L_J$$

$$M_1 = \frac{Q^2}{gA_1} + A_1 \overline{Y}_1 \cos\theta + \frac{A_1 + A_2}{2} cL_J \sin\theta$$

$$M_2 = \frac{Q^2}{gA_2} + A_2 \overline{Y}_2 \cos\theta$$

When the upper conjugate depth exceeds D_f then the resulting pressure head (H_{p2}) is calculated by:

$$H_{p2} = \frac{M_1 - M_f}{A_f}$$

where M_f and A_f are the momentum and cross sectional area at full barrel flow respectively.

Length of the Hydraulic Jump (L_J).

The length of a hydraulic jump is estimated by empirical factors obtained from field observations (King and Brater, Handbook of Hydraulics; US Bureau of Reclamation, Engineering Monograph 25, "Hydraulic Design of Stilling Basin and Bucket Energy Dissipators."

$$L_J = \beta Y_2$$

where β is a factor defined in Table 1 and Y_2 is the upper conjugate depth.

Channel Slope	Froude >= 4	Froude > 3	Froude > 2	Froude > 1
0.00	6.15	5.54	4.99	4.49
0.05	5.20	4.68	4.21	3.79
0.10	4.40	3.96	3.56	3.21
0.15	3.85	3.46	3.12	2.81
0.20	3.40	3.06	2.75	2.48
0.25	3.00	2.70	2.43	2.19

Table 1. Factors (β) for estimating the length of a hydraulic jump based on channel slope and Froude number.

HEAD LOSS AT THE INLET

FHWA Hydraulic Design Series No. 5, "Hydraulic Design of Highway Culverts."

Unsubmerged (form 1):

$$Y_u = Y_c + H_{vc} + D_t K\left[\frac{Q}{AD_t^{0.5}}\right]^M - \beta S_o$$

Unsubmerged (form 2):

$$Y_u = D_t K\left[\frac{Q}{AD_t^{0.5}}\right]^M$$

Submerged:

$$Yu = \kappa\left[\frac{Q}{AD_t^{0.5}}\right]^2 + \zeta + \beta S_o$$

Minor loss at the inlet:

$$h_i = K_i\left(\frac{V_i^2}{2g}\right)$$

RECONCILING THE HEADWATER AND TAIL WATER WITH CULVERT BOUNDARY CONDITIONS

Tail Water and the Culvert Outlet Boundary.

The culvert outlet boundary is the condition just inside the outlet. It is assumed that the hydraulic head at the outlet boundary (H_o) is equal to the depth of the tail water (Y_{tw}).

$$H_o = Y_o + H_{po} = Y_{tw}$$

A user option is provided that will prevent H_o from ever being less than one half the sum of the critical depth and the depth to the top of the barrel:

$$H_o \geq \frac{D_t + Y_c}{2}$$

Headwater and the Culvert Inlet Boundary.

"Headwater" is the water depth (Y_u) just upstream from the culvert headworks (inlet). The "Culvert Inlet Boundary" is the hydraulic head (H_i) just inside the culvert barrel. The head loss through the headworks (h_i) is the difference between the energy head upstream (E_u) and the energy head of the boundary (E_i).

The parameters of greatest interest to designers are usually the upstream depth (Y_u) and the inlet head loss (h_i). Two general methods are used for calculating this depth:

1. Calculate Y_u directly using empirical functions based on the flow rate (Q), the headworks configuration, and the type and size of culvert.

2. Calculate the head loss through the headworks (h_i), and add it to the depth of flow at the inlet boundary (Y_i).

Method 1.

FHWA Hydraulic Design Series No. 5, "Hydraulic Design of Highway Culverts" provides functions for calculating the upstream depth (Y_u) directly based on the flow rate (Q), the headworks configuration, and the size of the culvert. The value of Y_u determined by this method is referred to as the "Inlet Control" depth.

The head loss (h_i) can be estimated by the difference between Yu and the flow depth at the inlet boundary:

$$h_i = Y_u - Y_n$$

for a mild slope and

$$h_i = Y_u - Y_c$$

for a steep slope.

Method 2.

The direct calculation of the head loss through the headworks (h_i) is performed using the appropriate formulas. A backwater profile is calculated from the outlet to provide a value for Y_i at the inlet. The head loss (hi) is added to the depth at the inlet boundary (Y_i) to get the upstream depth (Y_u). The value of Y_u determined by this method is referred to as the "Outlet Control" depth.

If the backwater profile does not reach the inlet, than inlet control should occur. If the backwater profile does reach the inlet, then outlet control should occur.

Appendix B

Prototype Culvert Module

ROADWAY CULVERT MODULE

The ROADWAY CULVERT MODULE is a tool for analyzing the hydraulic characteristics of a roadway culvert system. A "Roadway Unit" unit contains the data defining the roadway geometry and the hydrology including flows and tailwaters. A "Culvert Unit" contains the data defining a culvert barrel and associated headworks. A Roadway Unit may contain more than one culvert unit.

Figure 1 shows the initial screen for the ROADWAY CULVERT MODULE. It contains four tabs across the top, and it is open to the first tab for the "Roadway" data. The remaining three tabs are for entering culvert data, and presenting the results in tabulated and graphical forms.

THE ROADWAY PAGE

The "Roadway" page contains three major sections. On the left is an open vertical panel used to identify the Culvert Units associated with this Roadway Unit. More than one Culvert Unit may be associated with a roadway.

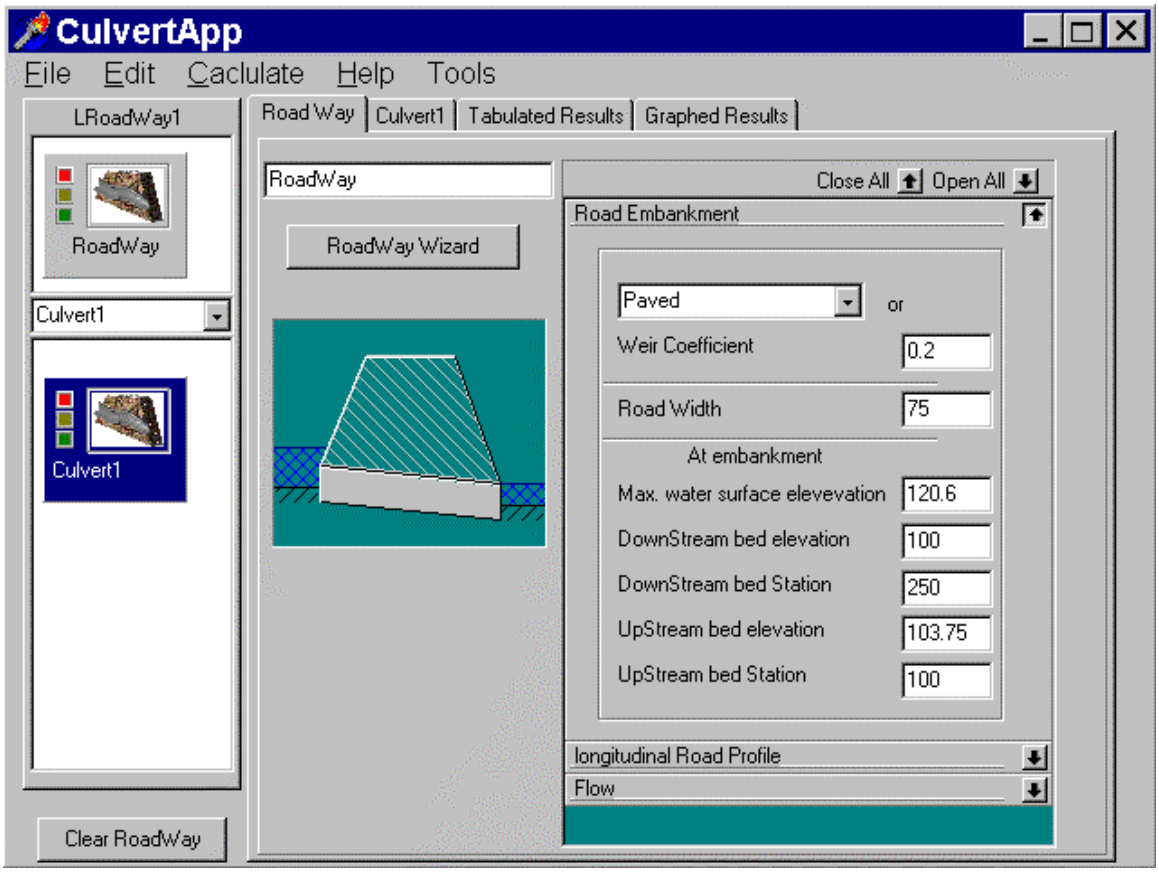


Figure 1. The initial screen for the ROADWAY CULVERT MODULE.

Roadway Input Data

The section on the right contains right in Figure 1 contains three horizontal panels for roadway data:

- "Road Embankment."
- "Longitudinal Road Profile."
- "Flow."

Roadway Embankment Panel. The "Road Embankment" panel is shown expanded in Figure 1. The following input data fields are provided on this panel:
Figure 1. The Roadway page with the "Road Embankment" panel expanded.

- A selection list for the type of road surface above the culvert. A "Weir Coefficient" is associated with the type of road surface. This coefficient is used to calculate the flow overtopping the road when the headwater elevation exceeds the elevation of the road. The user may directly input a value for this coefficient.
- The "Road Width."
- The "Maximum Water Surface Elevation." The model will flag a solution if the headwater elevation exceeds this limit. Future versions of the model will automatically calculate the minimum size culvert(s) needed to maintain the headwater elevation less than or equal to this "Maximum Water Surface Elevation."
- The "Down Stream Bed Station and Elevation." This is the station and elevation of the point where the downstream embankment intersects the streambed.
- The "Up Stream Bed Station and Elevation." This is the station and elevation of the point where the upstream embankment intersects the streambed.

The Flow Panel. Figure 2 shows the Roadway page with the "Flow" panel expanded. Two drop-down selection boxes are provided at the top of the panel for declaring the headwater and tailwater conditions. Three options are available for each:

- "Neglect." The energy of the approaching stream is neglected. The tailwater is low enough to prevent any downstream control.
- "Input." The user inputs values for depth and velocity for each flow.
- "Calculate." The user specifies the characteristics of the stream channel and the normal depth and velocity are calculated for each flow.

In Figure 2 the user has elected to neglect the energy in the approaching stream. He/she has elected to calculate the tailwater by specifying the characteristics of the channel downstream.

The Flow Panel. The "Flow" panel provides input fields for four flows ranging from the lowest flow of interest to the maximum flow of interest. Results will be calculated for each of these flows.

In addition a “Q_Inc” (flow increment) field is provided. Results will also be calculated at this increment from low flow to maximum flow. In this example results will be calculated for flows ranging from 150 cfs to 1575 cfs in increments of 100 cfs. Results will also be calculated at the four input flows.

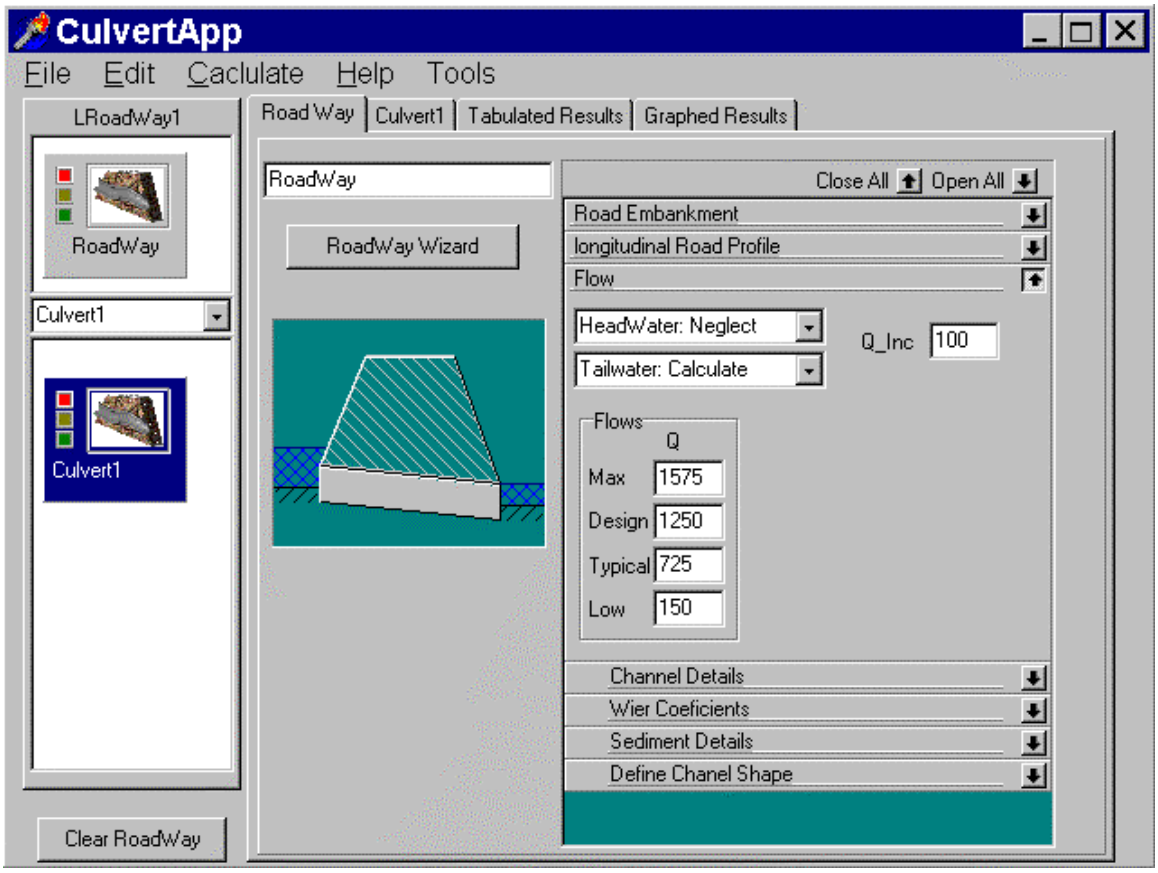


Figure 2. The Roadway page with the “Flow” panel expanded.

The Tailwater Channel Characteristics. The user elected to calculate the tailwater conditions by selecting “Tailwater: Calculate” in the drop-down selection box as shown in Figure 2. As a result, four panels appear to accept the data for the downstream channel: “Channel Details,” “Roughness Coefficients,” “Sediment Details,” and “Define Culvert Shape.”

The input data fields on these panels are similar to those in the FREE SURFACE FLOW MODULE. Figure 3 shows the Roadway page with the panels expanded for the downstream channel data that will be used to calculate normal depths for the tailwater elevations. A rounded triangle has been specified for the downstream channel. The geometric parameters are shown in Figure 3.

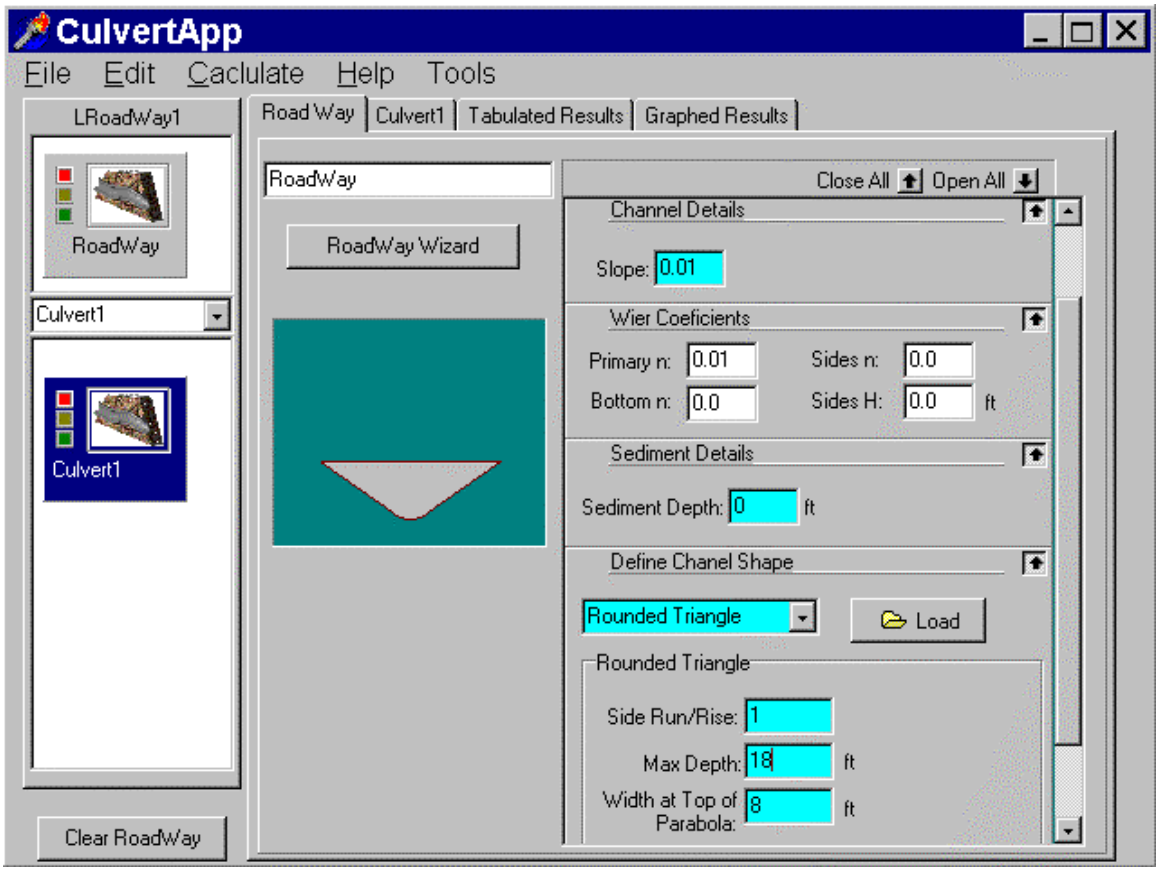


Figure 3. The Roadway page with the panels expanded for the downstream channel data.

THE CULVERT PAGE

Figure 4 shows the "Culvert" page with the "Define Culvert Shape" panel expanded. A 7'-2.25" x 5'-7.25' pipe arch has been selected form the file of standard shapes. This is the same section that is used in the example in the "FREE SURFACE FLOW MODULE" section of this Users' Guide.

Figure 5 shows the "Culvert" page with additional panels expanded for input data: "Roughness Coefficients," "Culvert Details," and "Sediment Details." These panels have been previously defined, and contain the same input data, as the example in the "FREE SURFACE FLOW MODULE" section of this Users' Guide.

Notice also that a data field is provided for the "Number of Barrels." The user may specify more than one barrel of this identical shape, slope and roughness for the "Culvert Unit" represented by this page. Two barrels have been specified in this example. The computer module simply proportions the flow among the barrels (e.g. 50% to each barrel in the example). Multiple barrels in a Culvert Unit are assumed to act independently – headworks conditions on one do not affect the headworks conditions on another.

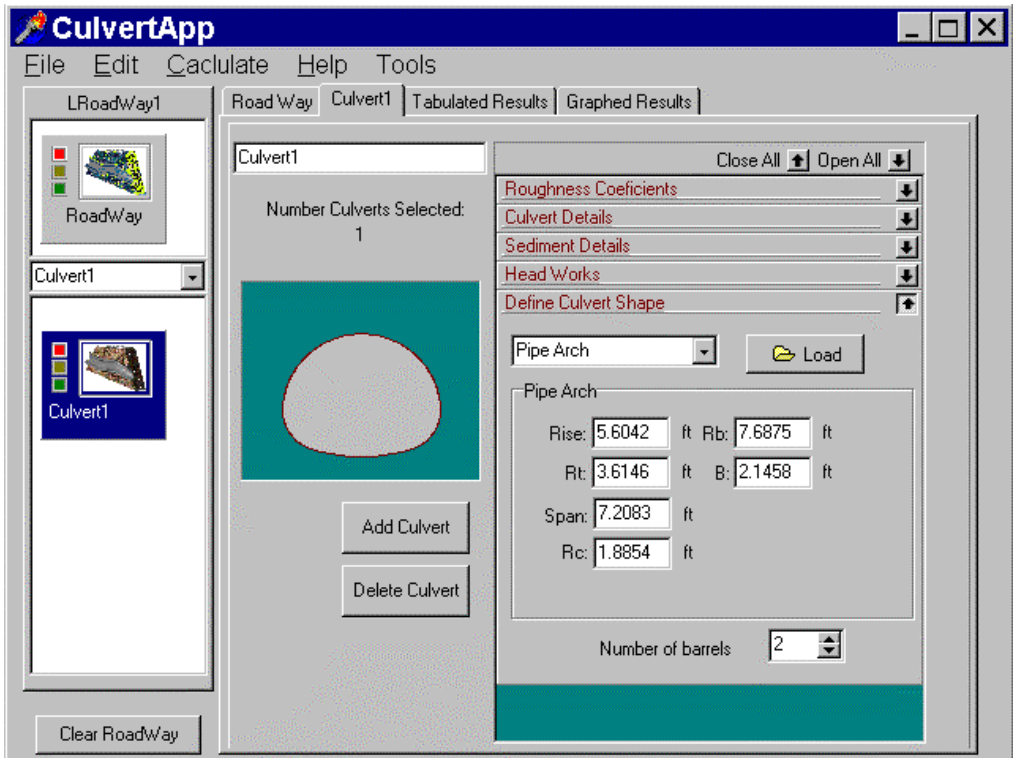


Figure 4. The Culvert Page with the "Define Culvert Shape" panel expanded.

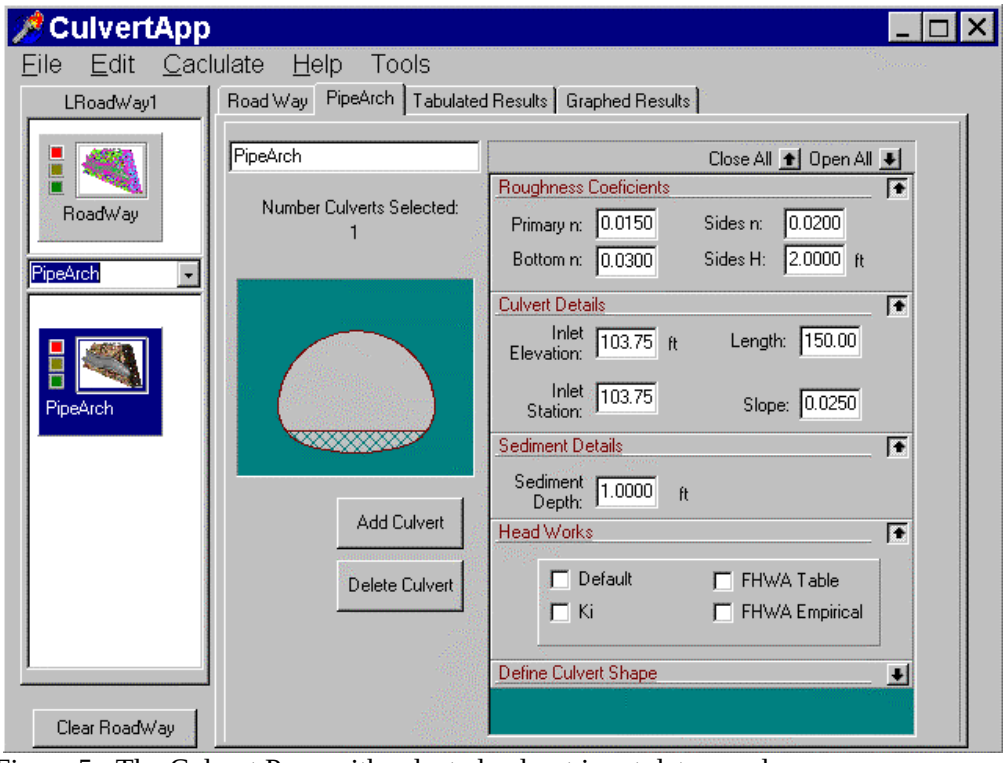


Figure 5: The Culvert Page with selected culvert input data panels open.

In addition to the data required to define the Culvert Unit, data are required to define the headworks associated with each culvert barrel. All barrels in a Culvert Unit must have the same headworks and the conditions at one headworks do not affect the conditions at another. The "HeadWorks" panel in Figure 5 allows the user to select one or more methods for calculating the headworks energy losses at the inlet.

- Default. Not defined in this version of the computer model and will probably be removed.
- Ki. A minor loss coefficient.
- FHWA Table. The formulas and tabulated coefficients in HDS-5.
- FHWA Empirical. The fifth order polynomial representations of the FHWA Table data.

Data fields are provided for the specific options selected by the user. For example, Figure 6 shows the panel expanded for the FHWA Table method. The coefficients for a pipe arch with no bevels have been loaded from the standard table by means of the "Load Table" button. This completes the input data needed for the Culvert Unit.

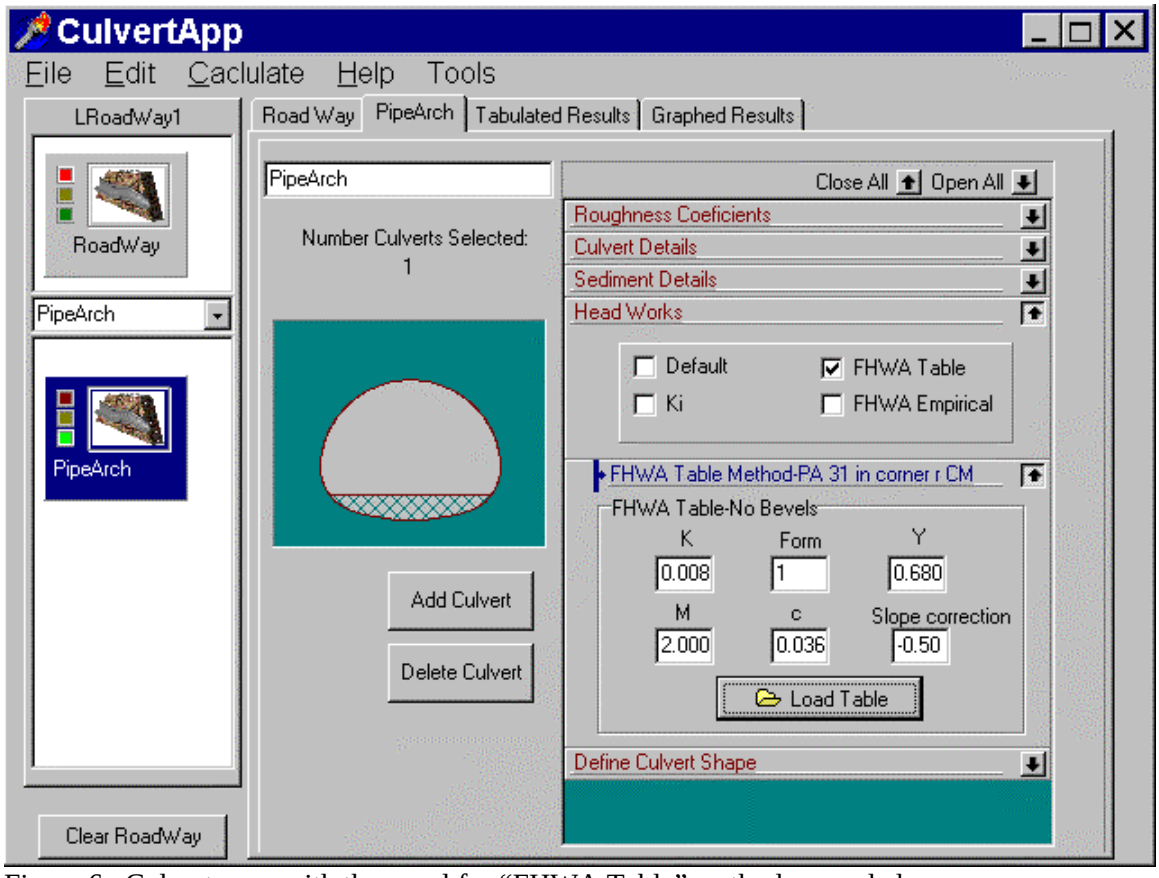


Figure 6. Culvert page with the panel for "FHWA Table" method expanded.

CALCULATE RESULTS

Figure 7 shows a pop-up menu containing the "calculate" item. This menu is obtained, after all data has been input, by clicking on the culvert icon with the **right** mouse button. Clicking on the "calculate" item will generate the output.

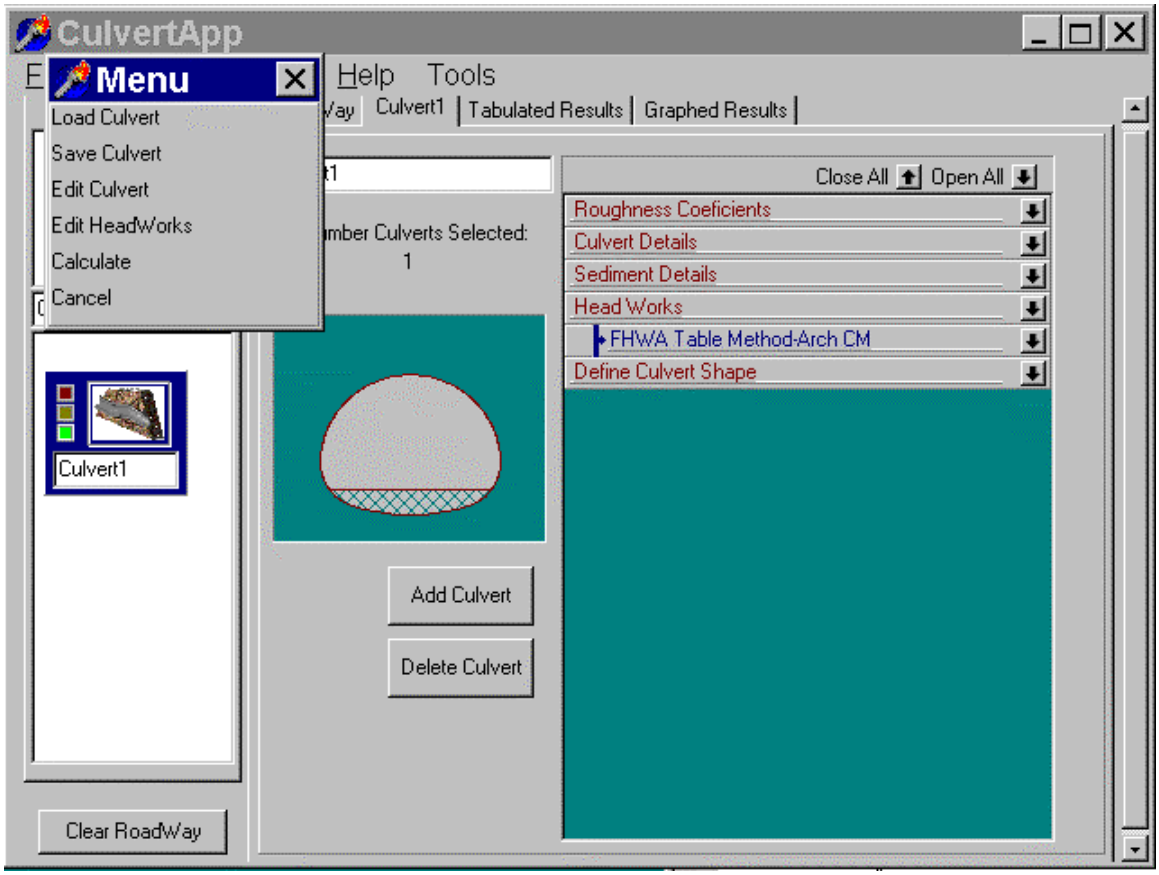


Figure 7. Pop-up menu for calculating the results.

Figure 8 shows the "Tabulated Results" page after the calculation. TotQ is the total flow, QPB is the Flow per Barrel, Ytw is the tail water depth, YupI is the upstream depth due to inlet control, and YupO is the upstream depth due to outlet control.

Figure 9 shows the "Graphed Results" page. The graph plots YupI and YupO versus total flow.

NOTE: THE CALCULATE FUNCTION, THE TABULATED RESULTS PAGE AND THE GRAPHED RESULTS PAGE ARE NOT PRODUCING THE CORRECT ANSWERS IN THE BETA 1.0 VERSION OF THE SOFTWARE.

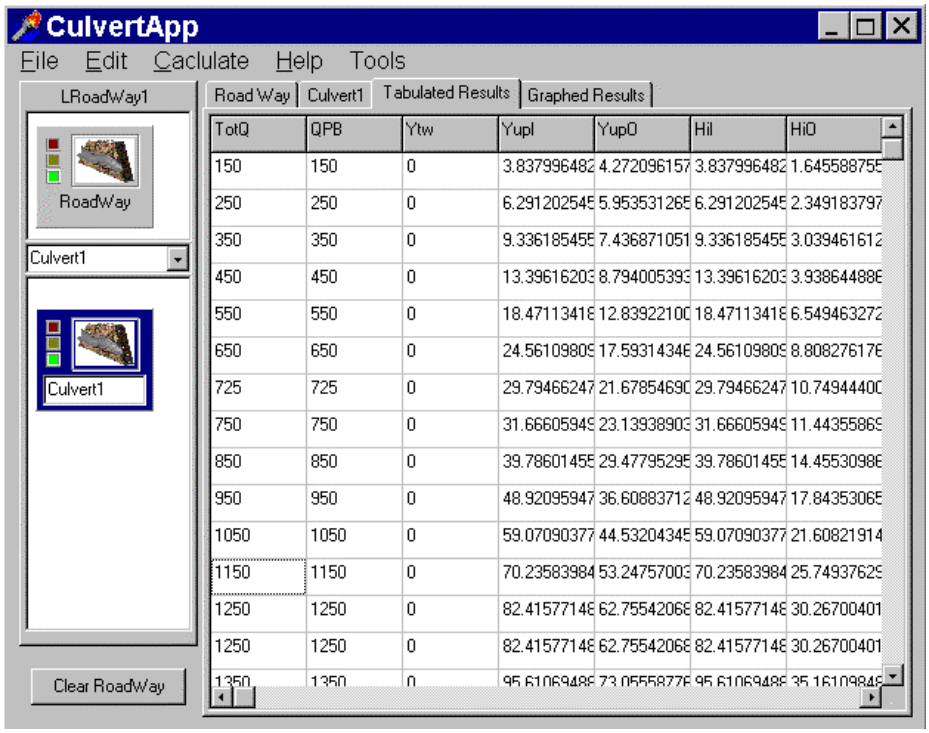


Figure 8: Tabulated Results.

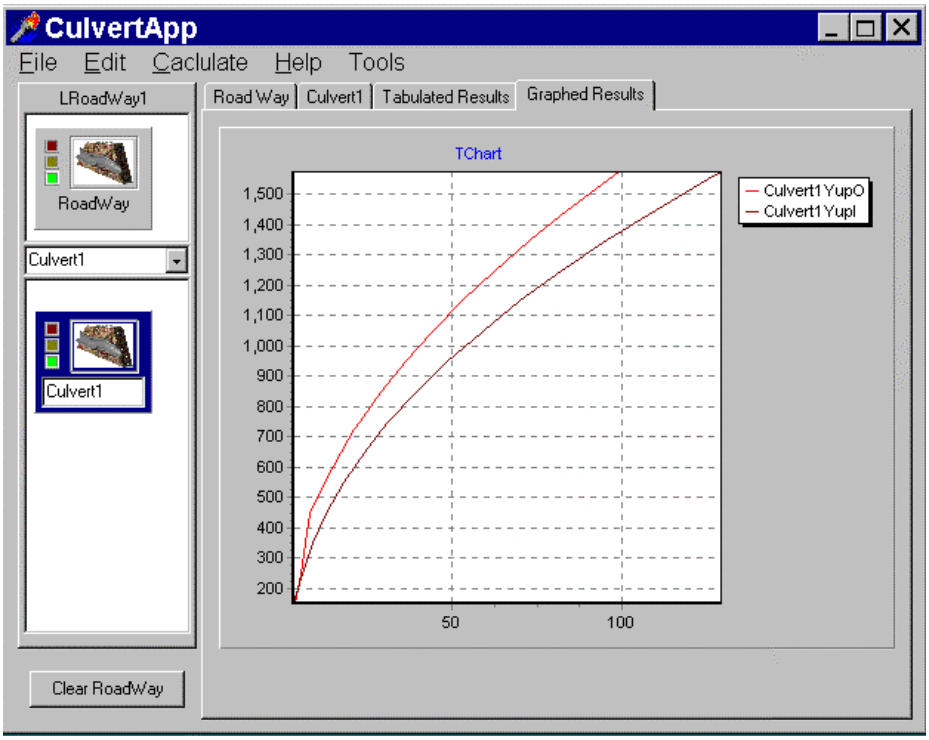


Figure 9. Graphed Results.

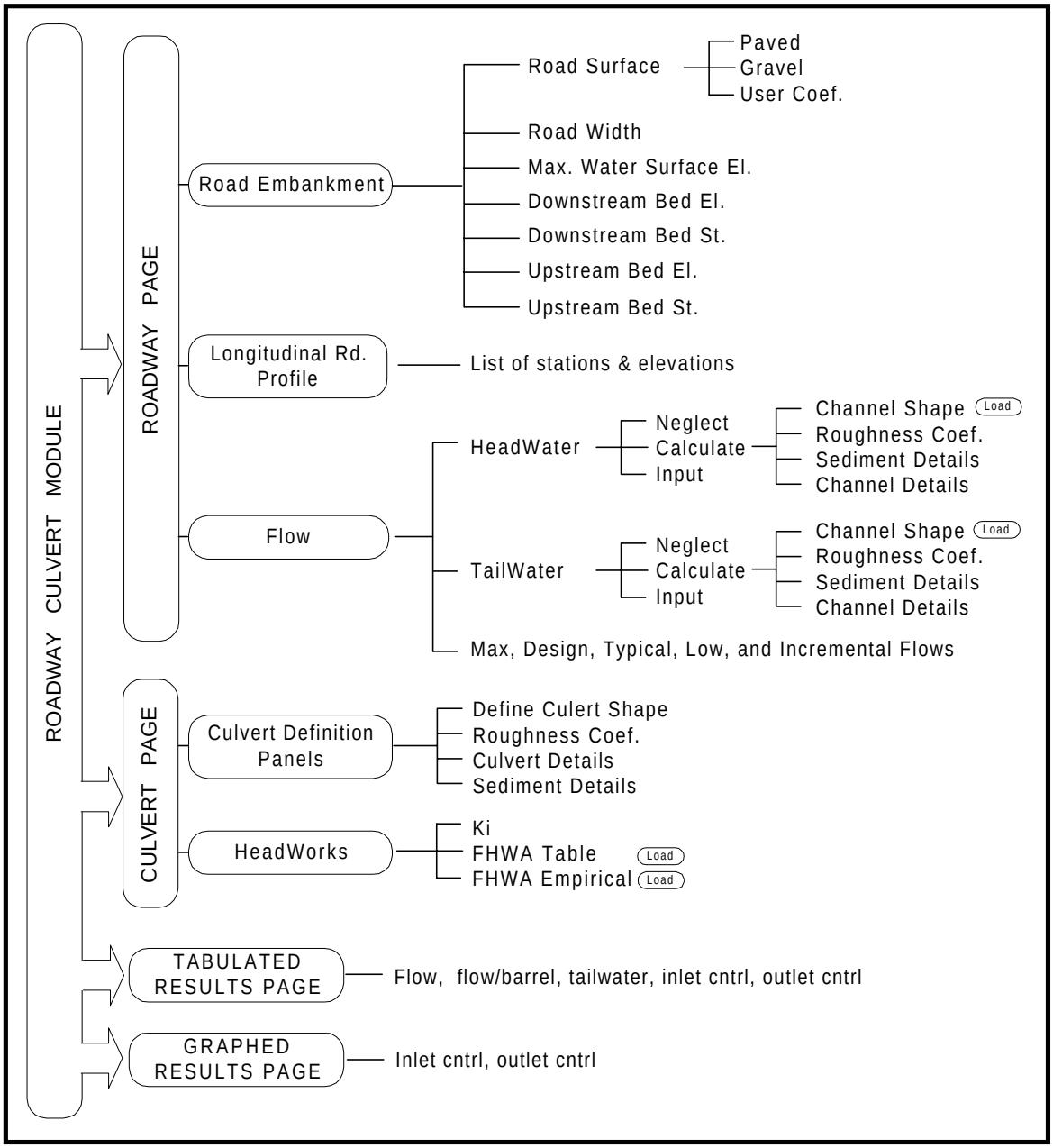


Figure 10. Roadway Culvert Module Architecture

FREE SURFACE FLOW MODULE

The FREE SURFACE FLOW MODULE is a tool for calculating the following information about flows in open channels and closed barrels commonly used for culverts:

- The shape geometry for 16 standard shapes.
- The hydraulic properties of a user defined shape as well as the standard shapes.
- The water surface profile for a specified upstream or downstream boundary condition.

OPEN CHANNEL HYDRAULIC PROPERTIES

Figure 1 shows the main screen for the FREE SURFACE FLOW MODULE. This screen contains two tabs at the bottom: one for calculating open channel hydraulic properties and the other calculating water surface profiles.

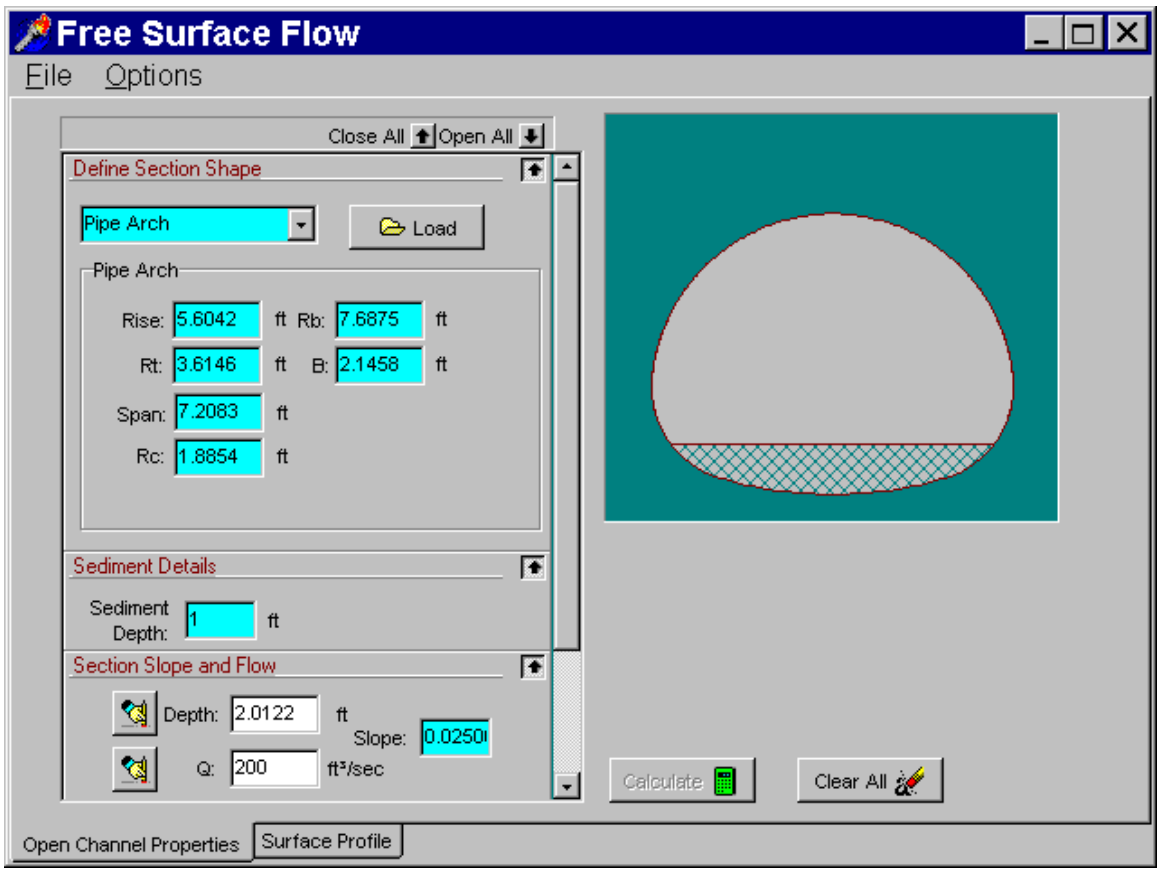


Figure 1: The main screen for the FREE SURFACE FLOW MODULE

Input and Results Panels

The Open Channel Properties page contains three sections: data input panels, results panels, and a graph panel for the cross section of the specified channel.

For example in Figure 1 a 7'-2.5" x 5'-7.25" pipe arch has been selected in the "Define Section Shape" input panel. The parameters defining the geometry for the section were loaded from a data file of standard shapes. A sediment depth of one foot has been specified in the "Sediment Details" input panel. The graph on the right shows the cross section and indicates the sediment depth.

The "Section slope and Flow" panel contains three fields. The slope of the channel goes into the "Slope" field. The remaining two fields are "Depth" and "Q" (flow). Only one of these fields may contain a non-zero value. If Q is specified (e.g. 200 cfs) then the depth must be set to zero (erased by the little button). When the "Calculate" button is clicked, then the normal depth will be calculated (e.g. 2.0122 ft). If the "Depth" is specified then the flow, Q, will be calculated for that normal depth.

Figure 2 shows the page after the "Define Section Shape," "Section Slope and Flow," and "Sediment Details" panels have been collapsed and the "Roughness Coefficients" and "Normal Flow Results" panels have been expanded.

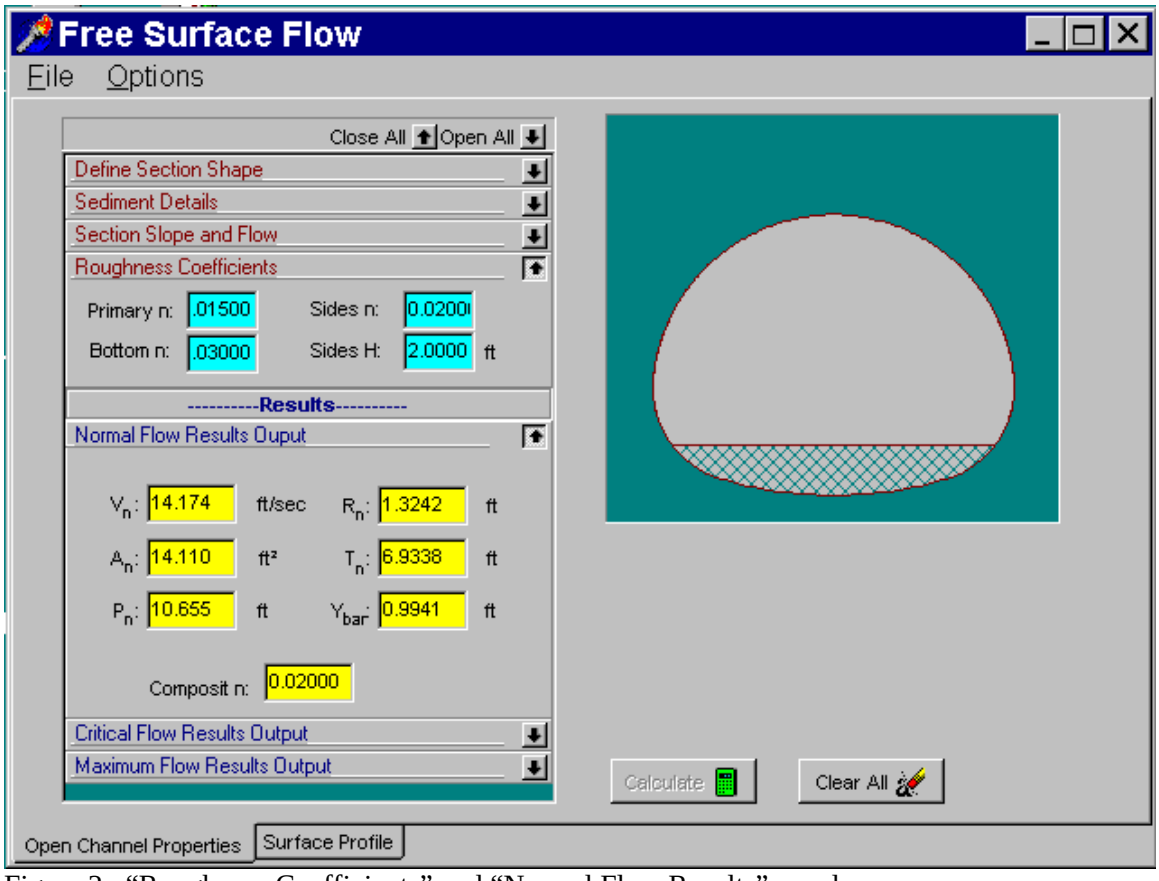


Figure 2: "Roughness Coefficients" and "Normal Flow Results" panels.

The "Roughness Coefficients" panel has input fields for three values of Manning's n. The "Primary n" is applied to the entire shape if non-zero values are not entered into the other fields. If a non-zero value is entered into the "Bottom n" field then this value replaces the "Primary n" for the flat bottom portion of the shape. If a non-zero value is entered into the "Sides n" then this

value replaces the "Primary n" for a distance up the sides equal to the value in the "Sides H" field.

The "Normal Flow Results" panel presents the normal flow properties including: normal velocity (V_n), wetted cross section area (A_n), wetted perimeter (P_n), hydraulic radius (R_n), top width (T_n), distance from the water surface to the centroid of the wetted area (Y_{bar}), and the calculated composite Manning's n value.

The normal flow, "Q" and normal depth "Depth" are presented in the "Section Slope and Flow" panel as shown in Figure 1.

Figure 3 shows the page after the "Roughness Coefficients" and the "Normal Flow Results" panels have been collapsed and the "Critical Flow Results" and Maximum Flow Results" panels have been expanded.

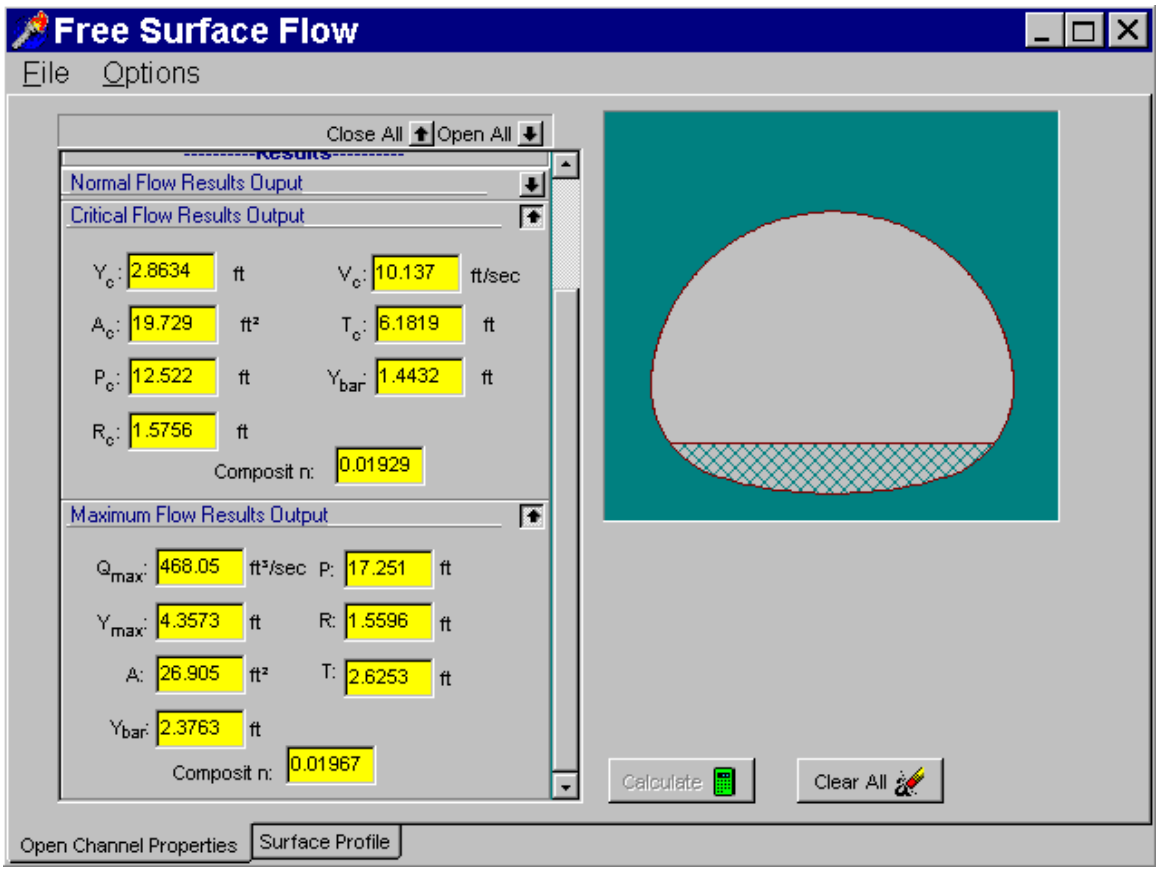


Figure 3. Critical Flow and Maximum Flow Results panels.

The "Critical Flow Results" panel presents the same properties for critical flow that were presented in the previous panel for normal flow. The critical depth is presented at the top of the list (Y_c).

The "Maximum Flow Results" panel presents the same properties for the maximum normal flow that can be carried by this section at the specified slope and with the specified Manning's n values. "Q_{max}," is the maximum flow without an upstream pressure head or downstream control. Notice that the depth of the maximum flow (4.3573 ft) is less than the depth of full barrel flow:

(full barrel depth) = (rise) – (sediment depth) = (5.6042) – (1.0) = 4.6042 ft

This will be the case for all closed shapes.

Functional Buttons.

The "Erase" button is used to erase the contents in the associated input field.

The "Calculate" button triggers the computational algorithms that read the input data and generate the results. When the "Calculate" button is red the input data has been changed since the last time the button was clicked. In other words, the current results were not generated from the modified input data. Red means, "Stop from using the results until this button has been clicked." The button will turn green when it is clicked. Green means, "Go ahead and use the results, they have been generated from the current input data."

The "Clear All" button clear all of the fields in all of the input and results panels.

The "Close All" button closes all panels.

The "Open All" button opens all panels.

The "Load" button opens a dialog box for accessing a file containing the geometric properties for standard shapes.

WATER SURFACE PROFILES

Figure 4 shows the "Water Surface Profile" page with the "Graphed Results" panel open. The algorithms calculate the water surface profile for the shape and flow conditions defined on the previous "Open Channel Properties" page. Either an upstream or downstream boundary may be selected. The water depth, hydraulic grade line and energy grade line are calculated and graphed.

The Graphed Results Panel

The "Graphed Results" panel has two sections. The lower half of the panel provides for the input data including fields for:

- Declaring either an upstream or downstream boundary.
- The boundary depth.
- The boundary velocity.

- The length of the channel (slope length).
- Minor loss coefficient.
- Option for specifying that a downstream boundary must be greater than or equal to one half the sum of the critical depth and the rise of the section.

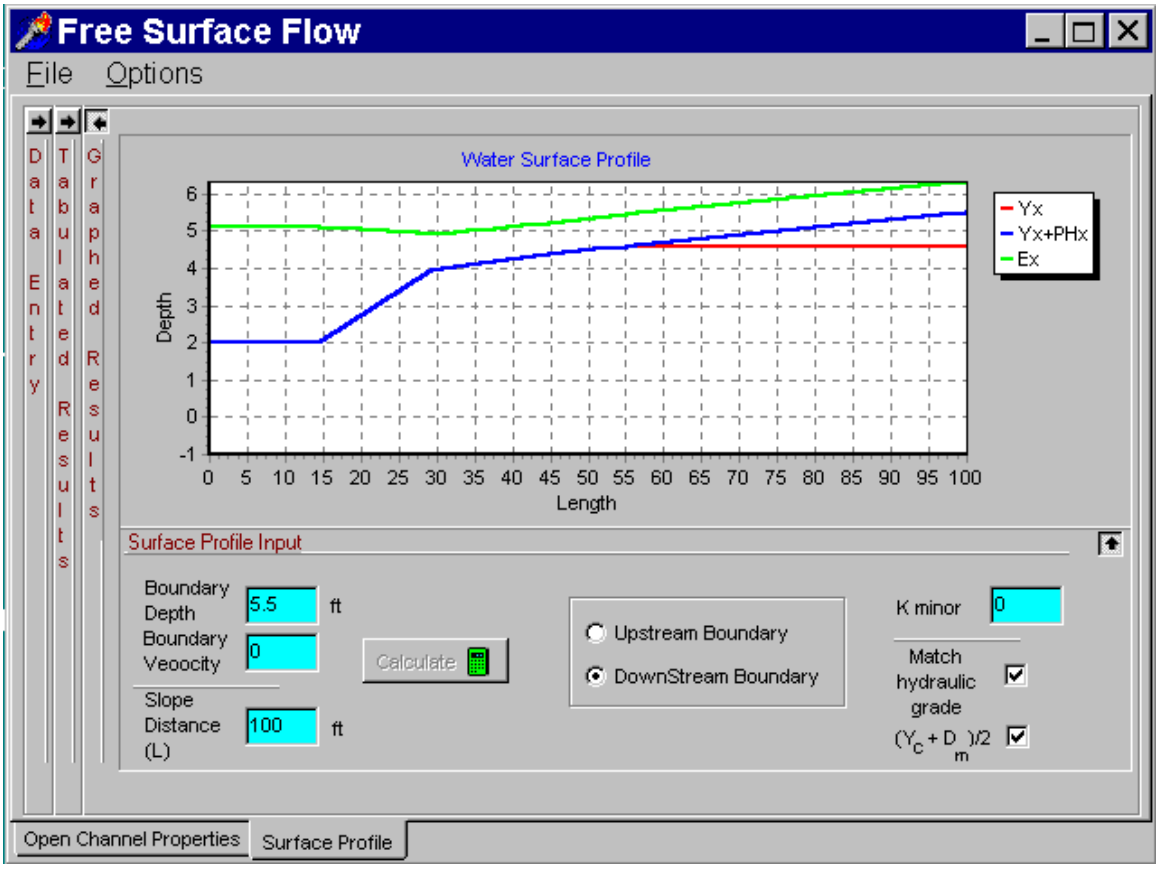


Figure 4. The Graphed Results panel on the "Water Surface Profile" page

The top half of the panel displays a graph of the water surface profile resulting from the shape and flow characteristics in the "Open Channel Properties" page, and the data values at the bottom of this page.

The graph in Figure 4 displays the results for the example shape and flow values previously entered in the "Open Channel Properties" page, and the values at the bottom of this page, namely: a downstream boundary depth of 5.5 feet, a length of 100 feet, match the hydraulic grade at the boundary, and require a downstream boundary not less than one half the sum of the critical depth and the rise of the section.

The plot shows the profile from the inlet (upstream) on the left to the outlet (downstream) on the right. The lower axis of the plot is aligned with the bottom of the channel – although we know that the channel bottom slopes down from left to right, our view is presented horizontally.

The channel (culvert) outlet is submerged. The top line extending the entire length of the channel is the energy grade line. The middle line is the hydraulic grade line. The lower line extending from about 55 feet downstream to the outlet is the depth of water at full barrel flow. The water surface profile can best be described by viewing the tabulated data along with the graphical display.

The Tabulated Results Panel

Figure 5 shows the "Surface Profile" page with the "Tabulated Results" panel open. The graph on the right automatically shrinks to fit the page. The "Tabulated Results" panel presents the following information:

$L_{x:}$ The slope distance from the inlet.

$Y_{x:}$ The water depth.

$PH_{x:}$ The pressure head.

VH_x: The velocity head.

E_x: The energy head (Yx + PHx + VHx).

The inlet (upstream) boundary is assumed to be at normal flow. The channel is on a steep slope and therefore Y_n (normal depth) is less than Y_c (critical depth) as displayed in the previous "Open Channel Properties" results panels. For the conditions in this example, downstream control will cause a hydraulic jump to occur in the barrel.

The first row in the table displays the input downstream boundary values. The second row displays the boundary condition just inside the outlet. Notice that a pressure head exists equal to the difference between the external boundary depth and the full barrel depth. Also a velocity head has been calculated for the depth at the outlet (full barrel in this example).

Starting at the outlet and moving upstream, the pressure head decreases until it intersects the top of the barrel at L_x = 55.1 feet. The depth of the water, represented by the middle line on the graph, is equal to the barrel depth while there is a pressure head. When there is no pressure head, the water depth coincides with the hydraulic grade. Free surface flow occurs at the point where the hydraulic grade intersects the top of the barrel, and a backwater curve extends upstream. The backwater curve intersects the upper conjugate depth of a hydraulic jump at L_x = 29.2 feet and Y_x = 3.95 feet. The jump extends to the lower conjugate depth, which is the normal depth, at L_x = 14.6 feet. Normal depth extends then to the outlet.

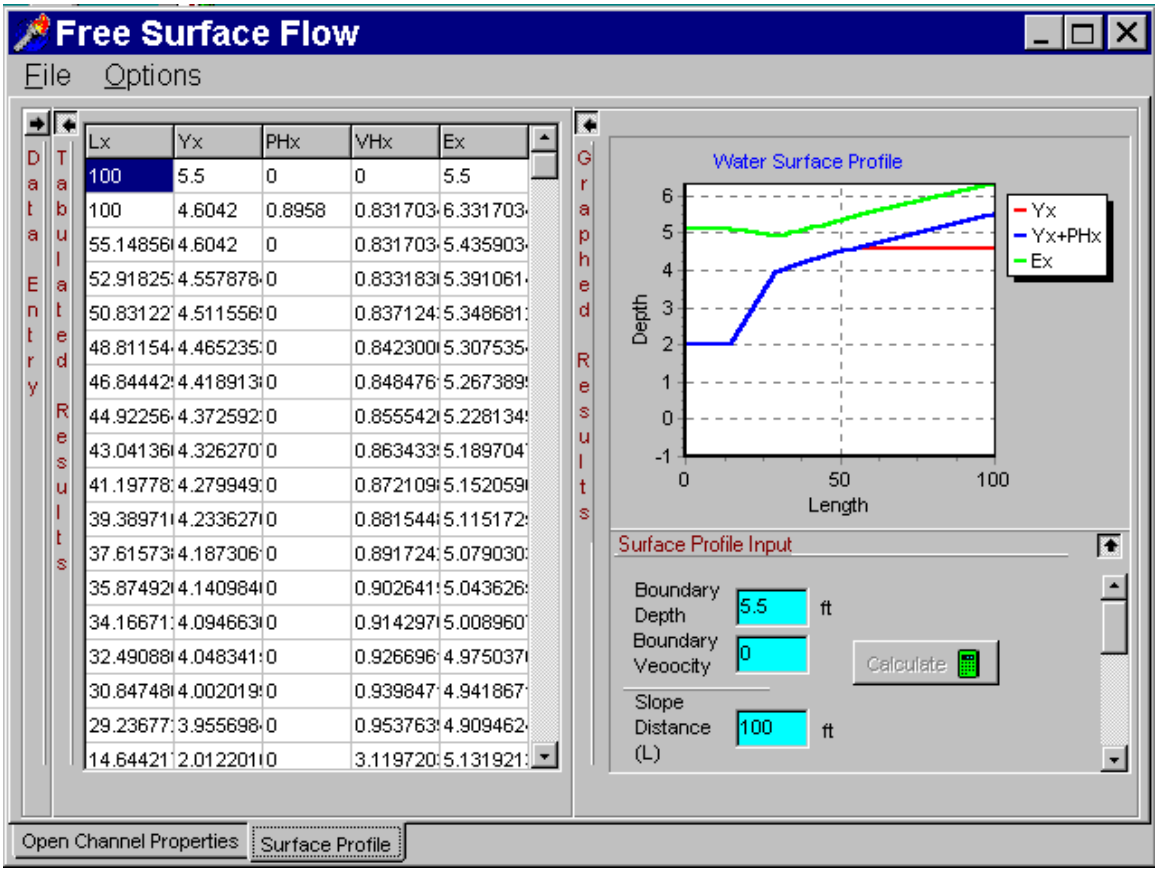


Figure 5. The "Surface Profile" page with the "Tabulated Results" panel open.

SUMMARY

Figure 6 is a schematic diagram of the interface structure for the FREE SURFACE FLOW MODULE. The module contains two main pages identified by tabs at the bottom: the Open Channel Properties page and the Water Surface Profile page.

The Open Channel Properties page contains seven horizontal panels that may be expanded or collapsed.

- **Define Section Shape**: Input data fields for specifying the channel/culvert geometry. The user can access standard shape files for six types of shapes by clicking on the load button.
- **Sediment Detail**: An input data field for the depth of sediment in the section.
- **Section Slope and Flow**: Input data fields for the channel slope and a range of flow values.
- **Roughness Coefficients**: Input data fields for three values of Manning's n: Primary n, Bottom n, and Sides n with associated distance up the sides from the bottom.

- **Normal Flow Results**: Output fields displaying the normal flow properties: depth (Y_n), velocity (V_n), wetted perimeter (P_n), hydraulic radius (R_n), water surface top width (T_n), distance from the water surface to the centroid of the wetted area (Y_{bar}), and the composite value of Manning's n (Comp-n).

- **Critical Flow Results**: Output fields displaying the critical flow properties: depth (Y_c), velocity (V_c), wetted perimeter (P_c), hydraulic radius (R_c), water surface top width (T_c), distance from the water surface to the centroid of the wetted area (Y_{bar}), and the composite value of Manning's n (Comp-n).

- **Maximum Flow Results**: Output fields displaying the properties for the maximum flow that can occur for this channel/culvert without an upstream pressure head: depth (Y_{max}), velocity (V_{max}), wetted perimeter (P_{max}), hydraulic radius (R_{max}), water surface top width (T_{max}), distance from the water surface to the centroid of the wetted area (Y_{bar}), and the composite value of Manning's n (Comp-n). For closed shapes, Y_{max} will be less than the rise of the shape.

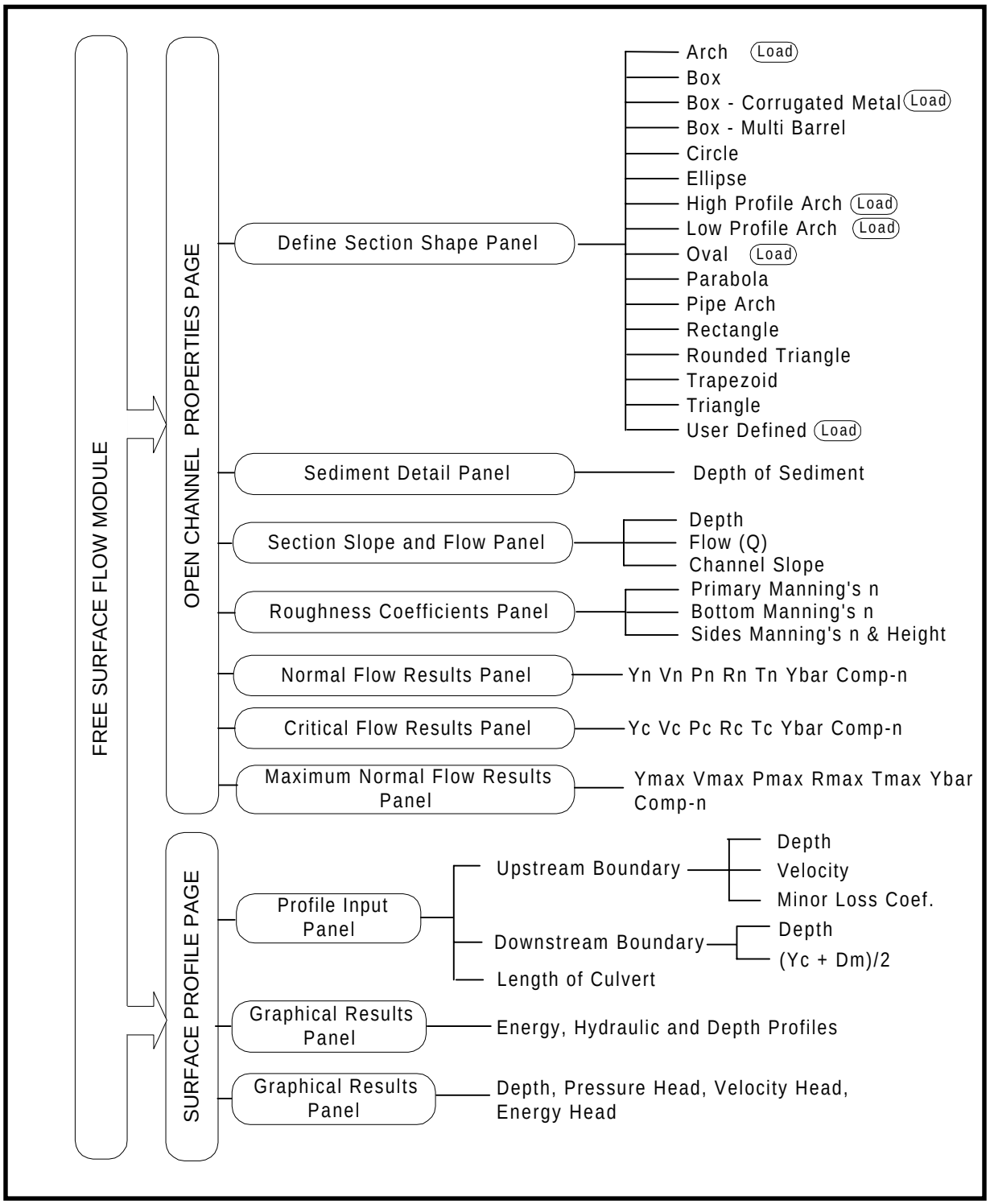


Figure 6. Schematic of the interface structure for the FREE SURFACE FLOW MODULE.

Appendix C

Models Removed During Initial Screening

Models Removed During Initial Screening

ANNIE

Interactive hydrologic and data management

ANNIE is a program designed to help users interactively store, retrieve, list, plot, check, and update spatial, parametric, and time-series data for hydrologic models and analyses. Data are stored in a direct access file called a Watershed Data Management (WDM) file. Many hydrologic and water-quality models and analyses developed by the U.S. Geological Survey (USGS) and the Environmental Protection Agency (EPA) currently use WDM files. The WDM file provides users with a common database for many applications, thus eliminating the need to reformat data from one application to another. There is also an expanding library of subroutines for graphics, user interaction, and data storage and retrieval available to application programmers designing software utilizing WDM files. (http://flash.net/~scitech/freesoft.htm)

AquaDyn

AquaDyn allows the complete description and analysis of hydrodynamic conditions (e.g., flow rates and water levels) of open channels such as rivers, lakes, or estuaries. (http://waterengr.com/)

BRANCH

The Branch-Network Dynamic Flow Model (BRANCH) is used to simulate steady or unsteady flow in a single open-channel reach (branch) or throughout a system of branches (network) connected in a dendritic or looped pattern. BRANCH is applicable to a wide range of hydrologic situations wherein flow and transport are governed by time-dependent forcing functions. BRANCH is particularly suitable for simulation of flow in complex geometric configurations involving regular or irregular cross sections of channels having multiple interconnections, but can be easily used to simulate flow in a single, uniform open-channel reach. Time-varying water levels, flow discharges, velocities, and volumes can be computed at any location within the open-channel network. Streamflow routing and computation by the BRANCH model is superior to simplified-routing methods in open-channel reaches wherein severe backwater and (or) dynamic flow conditions prevail. Typical uses of the model encompass the assessment of flow and transport in upland rivers in which flows are highly regulated or backwater effects are evident, or in coastal networks of open channels wherein flow and transport are governed by the interaction of freshwater inflows, tidal action, and meteorological conditions. Surface- and ground-water interactions can be simulated by the coupled BRANCH and USGS modular, three- dimensional, finite-difference ground-water flow (MODFLOW) models, referred to as MODBRNCH. (http://www.flash.net/~scitech/freesoft.htm)

CAP

The Culvert Analysis Program (CAP) follows USGS standardized procedures for computing flow though culverts. It can be used to develop stage-discharge relationships for culverts and to determine discharge though culverts from high water marks. It will compute flows for rectangular, circular, pipe arch, and other nonstandard shaped culverts. (http://www.flash.net/~scitech/freesoft.htm)

CHAN for Windows (Version 2.03)

CHAN for Windows is a modeling system that generates runoff hydrographs for basins and performs hydrodynamic routings of that runoff through a surface water system comprised of lakes, ponds, channels, and drainage structures. CHAN features all of the most popular methods for computing runoff and advanced hydrodynamic algorithms to provide fast stable simulations.
(http://www.aquarian-software.com)

CHANNEL

The CHANNEL program has six modules that automate design and analysis of open channels. It includes: HEC-15 flexible lining analysis, popup references that guide you through the input, context sensitive help, bends analyzed according to HEC-15 procedures, non-erodable linings that follow Ven Te Chow's text, cohesive and non-cohesive erodable analyses, best hydraulic section design, minimal lining cost design, and popup graphs to aid in soil parameter selection.
(http://www.waterengr.com)

Culverts (Part of Hydraulics Utility Programs)

In the Culverts program, the culvert characteristics are entered, and a headwater vs. outflow rating curve indicating inlet or outlet control is produced.
(http://waterengr.com)

DCUH

DCUH is a unit hydrograph program for the estimation of a unit graph from rainfall and discharge data.
(http://waterengr.com)

DR3M

The Distributed Routing Rainfall-Runoff Model (Version II) (DR3M) is a watershed model for routing storm runoff through a branched system of pipes and (or) natural channels using rainfall as input. DR3M provides detailed simulation of storm-runoff periods selected by the user. There is daily soil-moisture accounting between storms. A drainage basin is represented as a set of overland-flow, channel, and reservoir segments, which jointly describe the drainage features of the basin. This model is usually used to simulate small urban basins. Interflow and base flow are not simulated. Snow accumulation and snowmelt are not simulated. (http://www.flash.net/~scitech/freesoft.htm)

Flash20 (A pulldown version of TR-20)

Flash20 provides a user friendly data input program for SCS's TR-20 hydrology software. Generates a TR-20 input file from TR-55 hydrology data computed with Flash55. Allows you to graphically connect drainage areas and route them through channels and structures. Flash20 will automatically run TR-20 for you too, with no input required. And because you are still using SCS's TR-20 method, reviewing agencies will accept your data without the hassles associated with using other software.
(http://www.ramss.com/hydro.html)

Flash55 (A pulldown version of TR-55)

Flash55 is a hydrology program for PC-compatible computers. It performs SCS TR-55 method hydrology computation and report preparation with easy to use, pulldown menu user interface. Users can enter soil types, land uses, and areas, and the software selects the correct CN and percent of impervious area. The program also inputs elevations and flow lengths to automatically compute the watercourse slopes of Tc paths. An unlimited number of drainage areas, subareas, and Tc path segments are allowed. Total impervious area is also computed for each drainage area. Also transfers files from HYDROmate, TR-55 software for the HP48. (http://www.ramss.com/hydro.html)

Flow Pro 2.0

Flow Pro computes steady-state water surface profiles for many prismatic open channel shapes, including circular, rectangular, trapezoidal, triangular, u-shaped and tubular. It handles both subcritical and supercritical flow types, and includes several new tools for designing weirs, orifices, and underflow gates. Flow Pro also computes many useful flow and channel properties such as critical depth and slope, hydraulic radius and wetted perimeter, normal depth, and channel

roughness. It uses the Manning equation and numerical integration for state-of-the-art accuracy, and accepts both English and SI units of measure. (http://www.europa.com/~psapps/vault.htm)

Flow Tool

FlowTool is a hydraulic calculator program for PC-compatible computers. FlowTool allows the solution of equations relating to pipe pressure, pipe gravity, weir, orifice, gutter, and channel flows. All equations may be solved for any variable, and partial solutions are performed if not enough information is provided. A range for a variable may also be specified, and multiple solutions will be calculated automatically to generate rating tables. Also the program includes Pipe Analysis, which checks all flow types and determines the controlling factor. (http://www.ramss.com/hydro.html)

FLOWPROF

FLOWPROF computes one-dimensional steady state water surface profiles in prismatic and transition open channels.
(http://www.erols.com.cahh/)

FREQ

The program FREQ is a graphics-based LP3 flood frequency estimation program. Estimation is done by least squares. Features of the program are calculation of unbiases frequency factors and confidence limits for estimates. The program allows the user to alter the value of skew and/or peaks to years ratio. The effects of any changes can be monitired graphically.
(http://www.waterengr.com/)

GLSNET

GLSNET, a regional hydrologic regression and network analysis using generalized least squares, uses an analysis of residuals technique to estimate a regional regression equation to predict flow characteristics at ungaged sites. The regression analysis assigns different weights to observed flow characteristics. These weights are based on record length, cross correlation with flow characteristics at other sites, and an assumed model error structure.
(http://flash.net/~scitech/freesoft.htm)

Gully

Gully is a program to compute gully control structure design parameters. (http://www.waterengr.com/)

HEC-RAS

The River Analysis System (RAS) is the first of the Hydrologic Engineering Center's Next Generation Software programs. This Windows-based water surface profiles program will replace HEC-2 and ultimately, HEC-6 and UNET. (http://www.waterengr.com/)

HEC-1

HEC-1 is the U.S. Army Corps of Engineers flood hydrograph package for rainfall-runoff simulations. This program will produce runoff hydrographs for complex watershed networks incorporating reservoir and channel routing procedures. The program will allow various methods for calculating rainfall hyetographs, basin unit hydrographs, and watershed loss rates.

HEC-1v is a virtual memory hybrid version of HEC-1, developed by the Corps for evaluating extended hydrographs, with up to 1,000 data points. This can be invaluable for modeling storms of more than a few hours duration when small subareas, requiring short time steps, are involved.
(http://www.waterengr.com/)

HEC-5

The HEC-5 program is designed to simulate the sequential operation of a reservoir/channel system with a branched network configuration. Any time interval from one minute to a month can be used. Multiple time intervals can be used within a single simulation. Channel routing can be performed by any of seven hydrologic routing techniques. Reservoirs operate to: (1) minimize downstream flooding; (2) evacuate flood control storage as soon as possible; (3) provide for low flow requirements and diversions; and (4) meet hydropower requirements. Hydropower requirements can be defined for individual projects or for a system of projects. Pump storage operation can also be simulated. Sizing of conservation demands or storage can be automatically performed, using the safe yield concept. Economic computations can be provided for hydropower benefits and flood damage evaluation. Two editions of the HEC-5 package of programs are available: "overlayed" and "extended memory" (EM). While the basic programs are the same, the "EM' edition runs much faster and provides for 20 reservoirs and 40 control points, whereas the "overlayed" edition provides for 7 reservoirs and 15 control points.
(http://www.waterengr.com/)

HEC-2

This U.S. Army Corps of Engineers water surface profiles program has become the standard for FEMA floodplain evaluations and river channel design. The program runs in sub- or super-critical mode and has extensive bridge and culvert modeling capabilities. It will also model sideflow weirs, drop structures, and floodplain encroachments.
(http://www.waterengr.com/)

HEC-6

This program is used for the evaluation of scour and deposition in rivers and reservoirs. It is an extension to the HEC-2 channel geometry model, incorporating suspended and bedload sediment data. Sediment gradation and inflow tables are input into the model, as well as long-term inflow hydrographs. The output identifies reaches and depths of sedimentation and channel scour.
(http://www.waterengr.com/)

HEC-FFA

The HEC Flood Flow Frequency Analysis (FFA) program performs frequency computations of annual maximum flood peaks in accordance with the Water Resources Council "Guidelines for Determining Flood Flow Frequency," Bulletin 17B. This edition replaces the previous HEC-WRC program (date of last program release: February, 1995 (Version 3.1)). The program comes with a complete bound copy of the User's Manual. The program also comes with the U.S. Army Corps of Engineers' data storage system and GSS service drivers.
(http://www.waterengr.com/)

HSPEXP

HSPEXP is an expert system for calibration of the Hydrologic Simulation Program –Fortran (HSPF). It interactively allows the user to edit the input uci file for the HSPF, simulate with HSPF, produce plots of HSPF output compared to observed values, compute error statistics for a simulation, and provide the user with expert advice on which parameters should be changed to improve the calibration. In general, the user will spend time repeating the cycle of simulate, compute statistics, see plots, get advice, and edit the parameters.
(http://flash.net/~scitech/freesoft.htm)

Hydraflow Hydrographs

Hydraflow Hydrographs is a Windows-based rainfall-runoff and hydrograph development and routing model for complex watersheds.
(http://www.waterengr.com/)

HYDRAIN

The HYDRAIN program includes:

CDS – Culvert Design System

This program provides the engineer with the option of hydraulic design of a culvert or hydraulic analysis of an existing or proposed culvert. It routes hydrographs and considers both ponding and overtopping. The design option automatically selects a culvert size and the number of barrels, based on engineering data, environmental constraints, and site geometry. The review option will take the chosen culvert shape and size and will display hydraulic performance data for the selected culvert.

HY-8 – Culvert Analysis Program

Given the appropriate data, the program will compute the culvert hydraulics for circular, rectangular, elliptical, arch, and user-defined culverts. The logic involves calculating the inlet and outlet control headwater elevations for the given flow. The elevations are then compared, and the larger of the two is used as the controlling elevation. In cases where the headwater elevation is greater than the top elevation of the roadway embankment, an overtopping analysis is done in which flow is balanced between the culvert discharge and the surcharge over the roadway. In cases where the culvert is not full for any part of its length, open channel computations are performed.

HydroCulv

HydroCulv performs culvert hydraulic calculations to determine water surface profiles through culverts, based on culvert geometry data and boundary conditions. Output includes key results such as freeboard, head loss, inlet and outlet velocities, as well as depth and velocity profile information throughout the culvert. The output can be displayed in plots to aid in assessing the performance of the culvert over the range of boundary conditions and the sensitivity of results to certain variables.
(http://www.waterengr.com/)

HydroTech (V. 1.0)

HydroTech provides several scientific and engineering tools required in the analysis and manipulation of historical hydrological and water quality data. Statistical analysis tools and utilities include: log Pearson Type III, plotting position formulas, descriptive statistics, hydrograph data viewers, water quality indexing, and fisheries stream flow requirements.
(http://www.waterengr.com/)

HYRROM

HYRROM is an easy-to-use conceptual rain runoff model, with no requirement to understand the computing operating system or the structure of data files. The main use of a rainfall runoff model is to predict river flows from rainfall and evaporation data. In HYRROM, flows are predicted using a simple, realistic representation of the physical processes that govern water flow in a catchment. The model incorporates interception, soil, groundwater, and runoff stores, and includes some representation of the losses due to evapotranspiration. It can be calibrated manually or automatically using the built-in Rosenbrock optimization routine. The program is compatible with HYDATA. Output is in the form of color screen graphics that can be sent to a plotter or graphics printer, if required.
(http://www.scisoftware.com/)

Meltsum

Meltsum computes snowmelt from a degree-day model.
(http://www.waterengr.com/)

MLRP

The Multiple Linear Regression Program (MLRP) follows the procedures of "Statistical Methods in Hydrology" (Beard, 1962). Major features of the program are automatic deletion of independent variables (according to importance), combination of variables to form new variables, transformation of variables, tabulation of the residuals from the prediction equation, and acceptance of input coefficients. The program comes with a complete bound copy of the User's Manual.
(http://www.waterengr.com/)

PRMS

The Precipitation-Runoff Modeling System (PRMS) is a modular-design, deterministic, distributed-parameter modeling system developed to evaluate the impacts of various combinations of precipitation, climate, and land use on

streamflow, sediment yields, and general basin hydrology. Basin response to normal and extreme rainfall and snowmelt can be simulated to evaluate changes in water-balance relationships, flow regimes, flood peaks and volumes, soil-water relationships, sediment yields, and groundwater recharge. Parameter-optimization and sensitivity analysis capabilities are provided to fit selected model parameters and evaluate their individual and joint effects on model output. The modular design provides a flexible framework for continued model-system enhancement and hydrologic-modeling research and development.
(http://flash.net/~scitech/freesoft.htm)

RAMSS' Hydraulic Model

The RAMSS' Hydraulic Model performs channel, basin, and structure routings for complex networks of structures. It allows analysis of scenarios not possible using static rating table-based software. Multiple basin designs can be analyzed where varying tailwaters control discharges. Discharge types include circular pipe, box culvert, risers, weirs, and exfiltration.
(http://www.ramss.com/hydro.html)

SAC

The Slope-Area Computation (SAC) program follows USGS standardized procedure for computing discharge by the slope-area method.
(http://flash.net/~scitech/freesoft.htm)

SCSCN

The SCSCN is a storm runoff hydrograph model based on the SCS Curve Number method and NEH4.
(http://www.waterengr.com/)

SCSHYDRO

SCSHYDRO is a rainfall-runoff model based on U.S Department of Agriculture's Natural Resources Conservation Service (NRCS) (formerly the Soil Conservation Service (SCS)) hydrologic procedures. This model is a replacement for TR-20.
(http://www.erols.com/cahh/)

SMADA

SMADA for Windows is a complete hydrology program that includes a number of separate executable files. These programs work together to allow hydrograph generation, pond routing, storm sewer design, statistical distribution and

regression analysis, pollutant load monitoring, matrix calculation, and others. Extensive on-line documentation is available for all programs. These programs are useful in both classroom and professional applications.
(http://www.waterengr.com/)

STATS

The STATS program is designed to reduce large volumes of time series data to a few meaningful statistics or curves. The program will perform the following analyses: (1) duration curves; (2) annual maximum events; (3) annual minimum events; (4) departures of monthly and annual values from respective means; and (5) annual volume-duration exchange of high and low events. Data to be used as input to this program can be provided from a data file, or the data can be accessed through HEC's Data Storage System (HEC-DSS). This latter capability provides for the analysis of output from other HEC programs. The frequency analysis may be analytical for data that tends to follow a theoretical distribution, e.g., annual maximum daily flows; or graphical for those data that do not tend to follow any known distribution, e.g., regulated flows. The input data may be transformed to logarithms for the statistical analyses, if required. Program documentation consists of detailed input descriptions.
(http://www.waterengr.com/)

SWMM43

SWMM43 is the latest version of the Rainfall Runoff Model developed by the U.S. Environmental Protection Agency.
(http://www.waterengr.com/)

TOC

TOC computes overland time of concentration suing a combination of the Kinematic Wave Formula and the Green & Ampt Method. The technique was developed and published by Dr. Akan and received an ASCE award as an "outstanding contribution to irrigation and drainage research and education." In terms of formulation, this program is the state-of-the- art in time-of-concentration methods.
(http://www.erols.com/cahh/)

TR-20

TR-20 is the U.S. Department of Agriculture's Natural Resource Conservation Service (formerly the Soil Conservation Service) program for project formulation hydrology.
(http://www.waterengr.com/)

TR-55

TR-55 is the U.S. Department of Agriculture's Natural Resource Conservation Service (formerly the Soil Conservation Service) rainfall runoff model. The primary functions of the program are for peak runoff computations using the Graphical Peak Discharge Method, The Tabular Peak Discharge Method, and Temporary Storage. Support functions include the computation of the runoff curve (CN), the time of concentration (Tc), and travel time through a subarea (Tt). (http://www.waterengr.com/)

UNET

UNET (Version 3.1) was developed by the U.S Army Corps of Engineers. UNET simulates one-dimensional unsteady flow through a full network of open channels. In addition to solving the network system, UNET provides the user with the ability to apply several external and internal boundary conditions, including: flow and stage hydrographs; rating curves; gated and uncontrolled spillways; pump stations; bridges, culverts, and levee systems. The program comes with the Corps' Data Storage System software and GSS device drivers. UNET is written for DOS systems and comes with complete, bound copies of the User's Manual. (http://www.waterengr.com/)

Watershed Modeling

Watershed Modeling is a comprehensive drainage program that determines runoff using industry-standard techniques. The program models flood control structures such as detention basins with various outlet structures. Hydrographs can be created or imported from an ASCII file. Actual or synthetic rainfall distributions can be used. Hydrographs can be imported or combined at any point. The program can be used with Auto CAD, MicroStation, or as a stand-alone program. (http://www.eaglepoint.com/HH/HH.htm)

Water Surface Profiling

The Water Surface Profiling program uses HEC-2 and WSPRO methodology to compute flows or channels, bridges, culverts, etc. This model produces a comprehensive hydraulic analysis. It can produce plan, profile, and cross-section analyses.

(http://www.eaglepoint.com/HH/HH.htm)

Appendix D

Comparison of GEOPAK and Storm Works

INTRODUCTION

Numerous software packages are available for the design of storm drainage systems. These packages eliminate the tedium of having to design an entire network element wise. This is accomplished by designing a network from a single module. The power of engineering analysis along with interactive graphics helps reduce design and production time and increase quality control several notches. Keeping this end in mind, this study evaluated two such software packages: GEOPAK Drain and StormWorks. In preparing this report emphasis was placed on *what is there* to be done rather than *how it is to be done*. In other words the discussion details the features offered by each software package rather than the usage. In order to compare the software, knowledge about the constituent elements is a pre-requisite. Hence this Appendix is organized into sections that go into the salient features of GEOPAK Drain and StormWorks.

Chapter 1

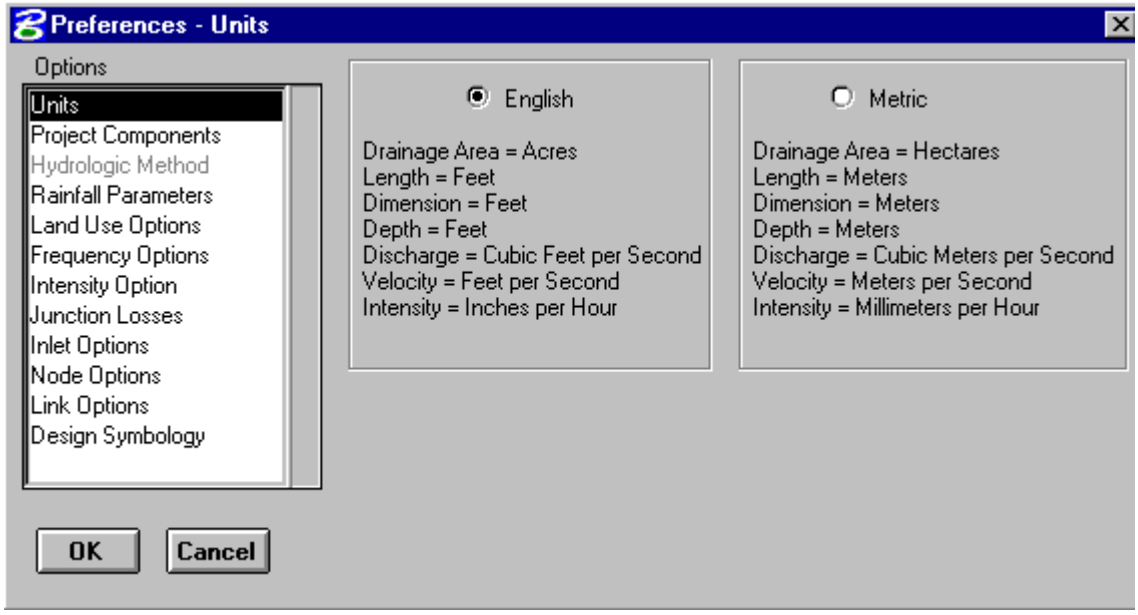
FEATURES OF GEOPAK DRAINAGE

PROJECT MENU

The Project Menu selections are utilized for creating new GEOPAK Drainage projects, opening existing projects, saving projects, establishing the project preferences and editing the Drainage Library. GEOPAK Drainage does not possess an auto save feature. Therefore, the Project > Save option must be regularly exercised.

PREFERENCES COMMAND

This option is invoked to support user-defined preferences that can be used to set project specific options. A dialog box offers preferences for various options. Below is a brief discussion of these options:



Units: A choice of English or Metric units can be made.

Project Components: Directories and files for Drainage Library, GPK file and Drainage Cell Library can be manipulated using this dialogue box.

Rainfall Parameters: Rainfall data source can be chosen.

Land Use Options: Drainage Library File is displayed and Land Use Data Source can be chosen.

Frequency Options: Computation frequencies and runoff coefficient peaking factors can be chosen.

Intensity Option: Minimum Time of Concentration and Intensity Options can be chosen.

Junction Losses: Establishes the default junction loss equations and coefficients utilized in pipe hydraulic calculations for the current project. Loss coefficients can be assigned to the following junctions:

Pressure expansion

Free surface expansion

Pressure contraction

Free surface contraction

Bend Loss computation

Terminal inlet/junction

Simple junction

Complex junction

Inlet Options: Inlet Bypass Options, Link Bypass Flow Options and Default Spread N values can be defined.

Node Options: Default Node ID Prefix and Automatic Link Updating option can be defined.

Link Options: The following items can be defined:

- Default Link ID Prefix
- Design optimization options: (a) minimize depth of cover; or (b) minimize pipe size
- Link Design Options: (a) design for full capacity; or (b) design partial capacity with [d/D] ratio input; or (c) design partial capacity with [q/Q] ratio input
- Hydraulic gradeline options: (a) equal hydraulic gradeline; or (b) equal energy gradeline.

Design Symbology: Element parameters for drawing both plan and profile views of drainage system are defined.

DRAINAGE LIBRARY

The Drainage lLbrary stores data and standards, which can be shared by different designers. There are 4 major categories in the Library:

- Rainfall Data
- Land Use Runoff Coefficients
- Nodes; i.e., inlets, outlets, junctions, etc.
- Links; i.e., pipes

Various File operations are provided in the Drainage Library. These allow one to create a new library, open an existing library file or save a file. For each of the above 4 categories, the user can add, delete or modify items associated with a particular category. Items included in each category are:

Rainfall Items: GEOPAK Drainage stores rainfall data either in the form of Intensity Duration Frequency (IDF) tables or as an equation. There are three different preset

intensity duration equation formats. For such rainfall equations, user-defined frequency variables can be input.

Land Use Items: These items are used to store runoff coefficients and graphic symbology for various land uses contained in drainage products. The parameters input are Item ID, which is used to recall the item from within a GEOPAK Drainage project, description of the land use item, description of each land use, runoff coefficients and level, color, style and weight of the graphic elements.

Node Items: Node library items contain descriptions, plan view representation and dimension information for node structures. The node types are curb, grate, slotted drain, junction, outlet and other generic nodes such as grade breaks. Further specifications for each node can be made, such as profile types, dimensions, etc.

Link Items: Link items are used to store the pipe configurations. Three types of properties are specified for each pipe:

(a) Shape: arch, box, circular, ellipse or pipe arch.

(b) Material: aluminum or steel.

(c) Type: standard, low profile or high profile.

When a pipe item is added or modified, a dialog box appears that prompts for the item ID, description, shape, material, Manning's roughness coefficient, rise and span.

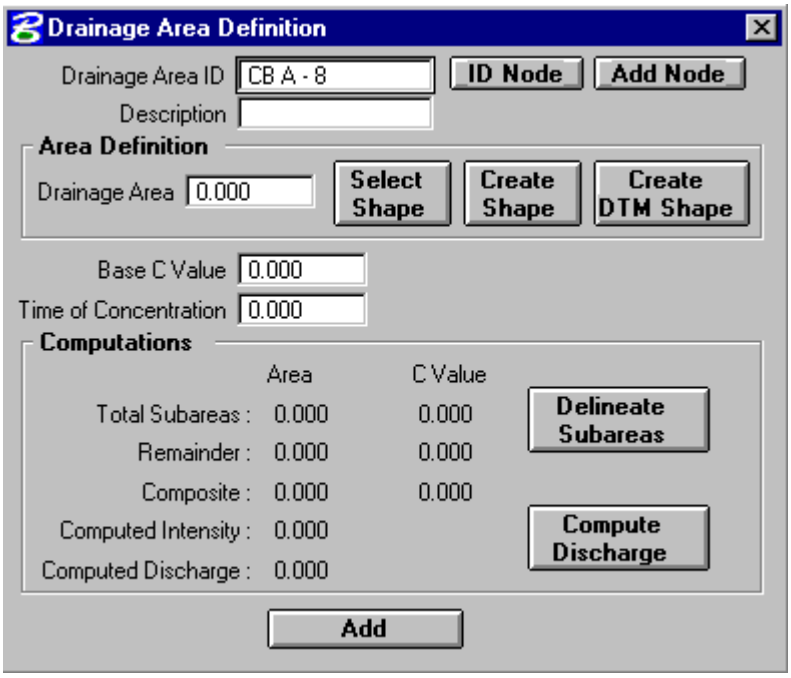
COMPONENT MENU COMMANDS

The component menu selections allow one to add, edit and delete the various elements that comprise a drainage system, which include Areas, Nodes, Links, Reaches, and Land

Uses. Each of these tools invokes a dialog wherein the specific component information can be added or edited.

DRAINAGE AREA ELEMENT

Drainage Areas are used to compute peak discharges or to attach computed discharge values to nodes within a project. The Drainage Area boundaries can be delineated using a Digital Terrain Model (DTM) drawn using MicroStation, or keyed in as a total area value. Using the commands in the component menu, drainage areas can be added, deleted, and edited. This dialog can also be invoked using the Navigator Tool. The Drainage Area Definition dialog box appears when areas are to be added, updated or deleted. The figure below shows the dialog box.



The dialog fields in the Drainage Area Dialog box with brief descriptions are:

Drainage Area ID: Drainage Area Identification

ID node: Enables the designer to select a previously defined node.

Add/Edit node: Enables the node configuration dialog box.

Description: This input is optional.

Drainage Area: This parameter is either input or is computed when a shape is selected.

Select Shape: Select a previously drawn shape.

Create Shape dialog: Create a new shape.

Create DTM shape: Create a Drainage Area shape using the Delineate Watershed-Drainage Area tool.

Base C value: The runoff coefficient for the drainage area.

Time of Concentration: Tc expressed in minutes.

When these values are input the following computations are performed:

Total Subareas and C value

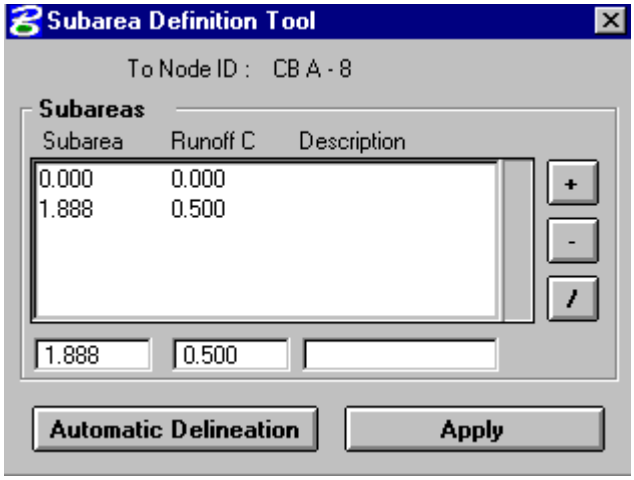
Remainder Area and Remainder C value

Composite Area and C value

Computed Intensity

Computed Discharge

GEOPAK Drainage allows one the flexibility to delineate Subareas. Subareas are smaller components within a drainage basin that exhibit different land coefficients or roughness coefficients. The fields input in this dialog box are the area of the Subarea, runoff coefficient and description. When the Automatic Delineation option is exercised, GEOPAK Drainage automatically delineates the subareas, drawing its values from the Land Use Table of the Drainage Library.

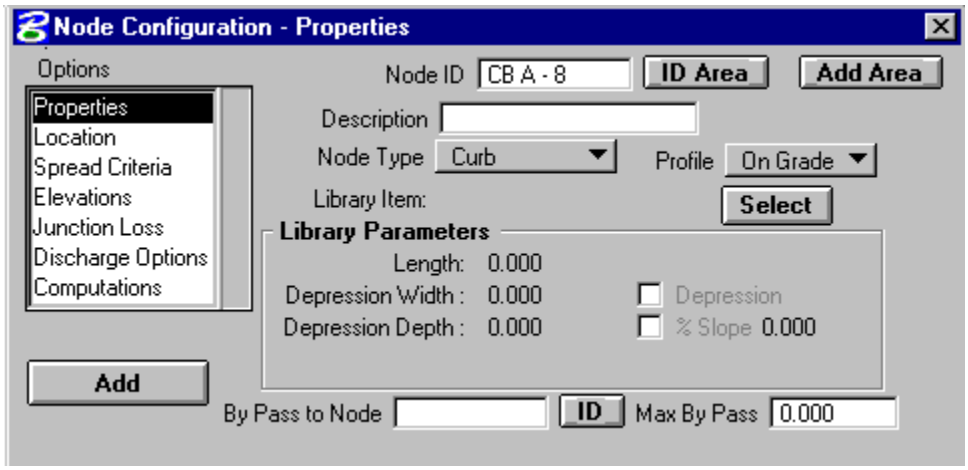


THE NODE ELEMENT

Nodes in GEOPAK Drainage are the key points within a drainage network. Nodes create the inlets, junctions and outlets in the network and provide for connectivity of the link system. Links can change size or slope only at nodes. Six categories of nodes may be placed in a drainage project, including:

- *Curb:* used for curb opening inlets with or without pipe confluences.
- *Grate:* used for grate opening inlets with or without pipe confluences.
- *Slotted Drain:* used for slotted drain inlets with or without pipe confluences.
- *Junction:* used for any confluence of pipes, structure location (that is not an inlet), pipe size or slope change, or flow addition that does not require an inlet.
- *Outlet:* used for the outlet, or outfall, of a storm drain network.

Using the Node pull-down menu, nodes can be added, edited or deleted. Selecting any of these options activates the Node Configuration Dialog box.



Various parameters can be specified for a variety of options that include:

Properties: Defines the type of node, library reference for the node and if applicable, bypass flow properties.

Location: Defines the location and reference information for the Node.

Spread Criteria: Applicable to inlets only, this option is used to define the spread criteria for ponded width and depth calculations used in the inlet hydraulic computations.

Elevations: Defines the Node elevation, critical elevation, and criteria for the elevations of Links connecting to the Node.

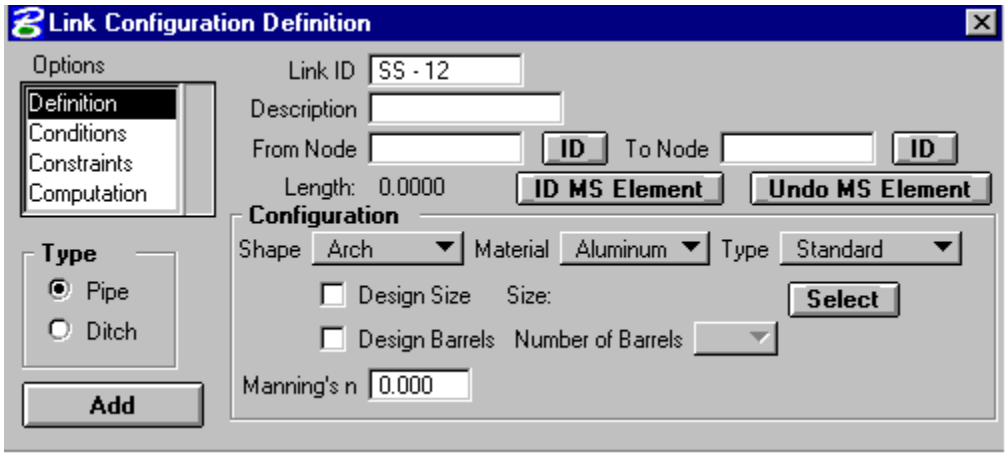
Junction Loss: Defines, overrides, or disables Junction Loss computations for the Node.

Discharge Options: Defines the Node discharge options and source of the discharge for the Node.

Computations: Applicable to inlets only, this option evaluates the hydraulic adequacy of the Inlet and displays the current hydraulic computations for the inlet.

THE LINK ELEMENT

Links connect and convey runoff from various nodes within a network to the outlet and consist of pipes, boxes or ditches. Links can be added, modified and deleted using the Link configuration definition dialog box.



Various sets of parameters can be chosen for four items as follows:

Definition: Defines the Link's path, connectivity, shape, material, roughness coefficient, size, and design options.

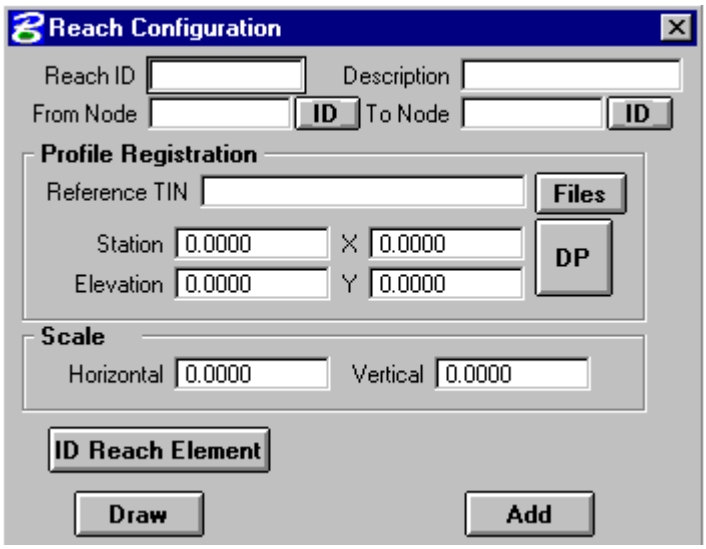
Conditions: Defines or reviews the elevation condition for the Link, including minimum depth, soffit, invert, and slope data. It also includes the profiling options for holding certain values constant.

Constraints: Defines the Link constraints, including the minimum and maximum rise (size), slope, and velocity used in Link design.

Computation: Reviews the Link hydraulic computations.

THE REACHES ELEMENT

A Reach is a path between two nodes, spanning one or more links. Reaches allow users to define a path running in either upstream or downstream direction in a drainage network and visualize profiles (ground, pipes, depth of cover, etc.) along that path. The identification of the From Node and To Node is required to define a Reach. GEOPAK Drainage automatically traverses the Network and finds the Links connecting the two Nodes. A Reach is only needed for the purpose of profile visualization and is not required to complete the drainage design process. Using the menu options provided, Reaches can be added, edited or deleted.

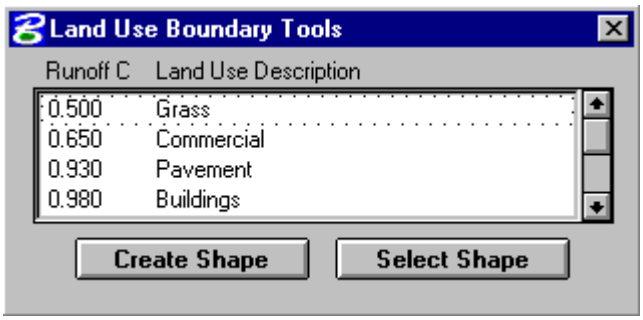


THE LAND USES ELEMENT

GEOPAK Drainage graphically defines areas or regions of different land uses and subsequent runoff characteristics in order to automatically track and compute runoff

coefficients for Drainage Areas. These land uses are defined using MicroStation shapes, specific symbology, and related runoff coefficient parameters.

When Component > Land Use is selected, a dialog box appears that consists of the following elements :



Runoff/ Land Use Description List Box: This box depicts the Land Use Descriptions and corresponding runoff coefficient values for entries in the currently selected Land Use Item within the Drainage Library.

Create Shape Button: This button is used to define a new Land Use Area.

Select Shape Button: This button is used if the Land use area has been previously defined as a MicroStation closed element.

NETWORKS MENU

A GEOPAK Drainage Network is defined as a series of interconnected Nodes and Links draining to a single outlet. Multiple Networks can be maintained in a single project. The final process in the design and analysis of a storm drain system are the network computations. Hydraulic computations on pipes and ditches require a network to be defined and successfully built. There are six options supported by the network pull-down menu:

Add: Used to define the network and associated data.

Edit: Utilized to edit a previously defined network.

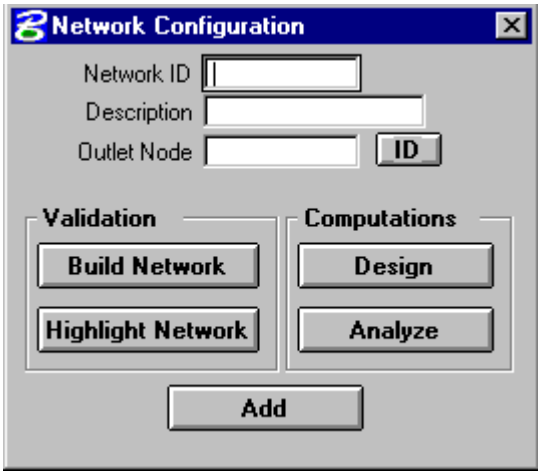
Delete: Utilized to delete a network and associated data.

Design: Enabled only when the active network is defined. Computes the active network system hydraulics and displays the number of nodes and links.

Analyze: Enabled when an active network is defined. Is a shortcut to the Analyze Procedure without having to invoke the Network dialog.

Active Network: Used as a shortcut and organization tool to limit the information viewed only to the active network.

When the add, edit and delete options are exercised, the network configuration dialog box appears whose features include:



Input Parameters

Network ID: Identification of the network.

Description: Description of the network.

Outlet Node: Identification of the outlet node.

ID: This button is used to dynamically identify the outlet node.

Validation Group Box

Build Network: This button builds the storm drain network starting from the outlet node. The purpose of building a network is to determine whether the storm drain system is ready to be analyzed or designed.

Highlight Network: This button highlights all elements, Areas, Nodes and Links contained in the active network to aid visual inspection.

Computations Group Box

Design: Performs the hydraulic design of all the components contained in the network and evaluates it.

REPORTS MENU

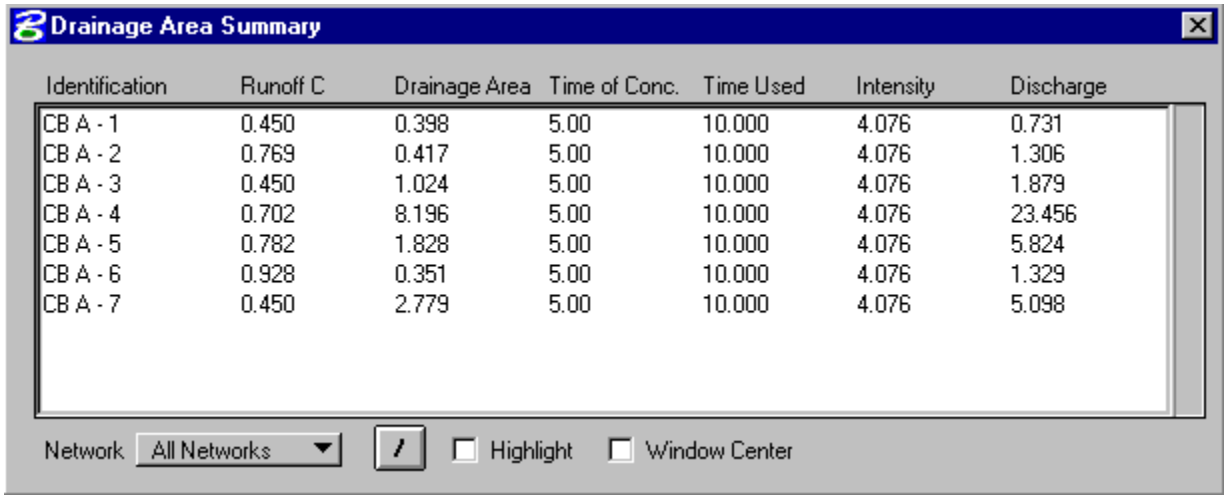
GEOPAK Drainage provides a set of commands to generate either custom or standard reports. The reports pull-down menu offers several report options. Of these, Drainage Areas, Inlets, and Storm Drains/Links are standardized reports, while the Builder and Generate menu selections are utilized for customized reports. Standardized reports are designed for viewing only, whereas hard copy output can be produced from customized reports.

A few features are common to all standardized reports. These features in brief are:

- Selecting a Network, either the Active Network or All Networks.
- An Edit option that brings up the Drainage Definition Dialog box.
- A Highlight option that is used to highlight a specific item.
- A Window Center option that centers the specific area selected.

The following screen shots show the reports along with the features afforded by them.

Reports > Drainage Areas

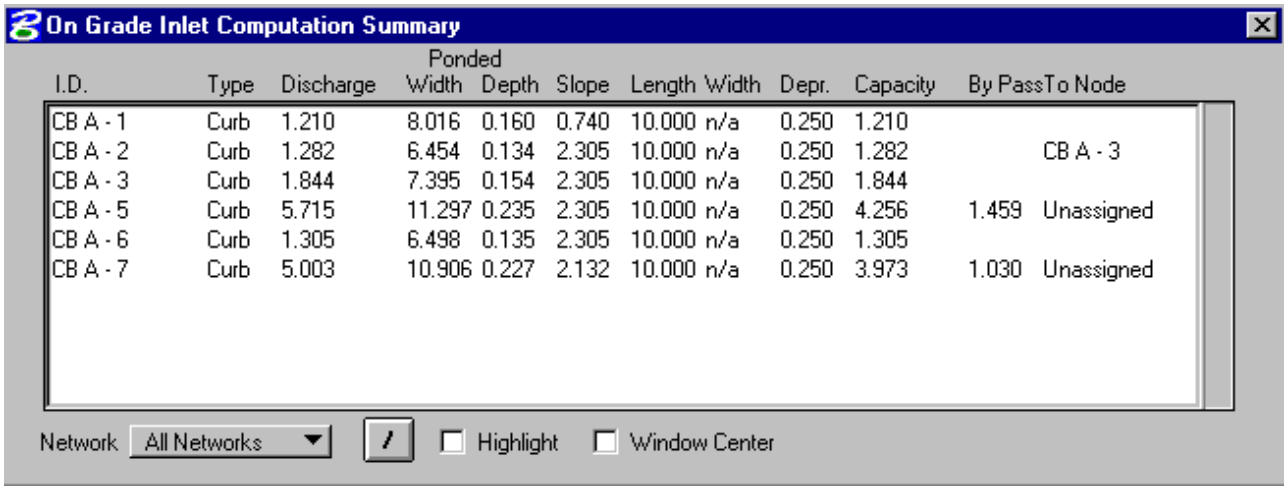


Drainage Area Summary

Identification	Runoff C	Drainage Area	Time of Conc.	Time Used	Intensity	Discharge
CB A - 1	0.450	0.398	5.00	10.000	4.076	0.731
CB A - 2	0.769	0.417	5.00	10.000	4.076	1.306
CB A - 3	0.450	1.024	5.00	10.000	4.076	1.879
CB A - 4	0.702	8.196	5.00	10.000	4.076	23.456
CB A - 5	0.782	1.828	5.00	10.000	4.076	5.824
CB A - 6	0.928	0.351	5.00	10.000	4.076	1.329
CB A - 7	0.450	2.779	5.00	10.000	4.076	5.098

Network All Networks / Highlight Window Center

Reports > Inlets > On Grade/On Sag Inlets

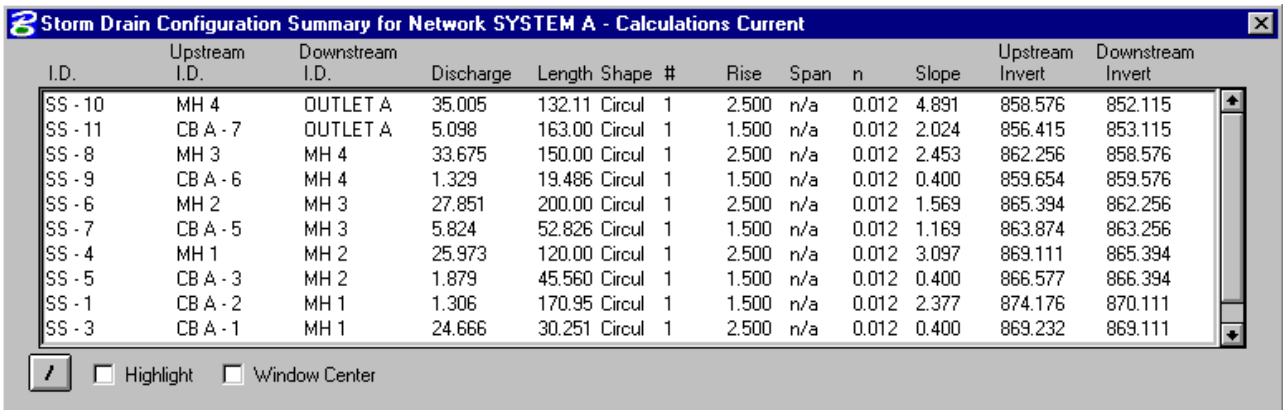


On Grade Inlet Computation Summary

I.D.	Type	Discharge	Ponded Width	Depth	Slope	Length	Width	Depr.	Capacity	By Pass	To Node
CB A - 1	Curb	1.210	8.016	0.160	0.740	10.000	n/a	0.250	1.210		
CB A - 2	Curb	1.282	6.454	0.134	2.305	10.000	n/a	0.250	1.282		CB A - 3
CB A - 3	Curb	1.844	7.395	0.154	2.305	10.000	n/a	0.250	1.844		
CB A - 5	Curb	5.715	11.297	0.235	2.305	10.000	n/a	0.250	4.256	1.459	Unassigned
CB A - 6	Curb	1.305	6.498	0.135	2.305	10.000	n/a	0.250	1.305		
CB A - 7	Curb	5.003	10.906	0.227	2.132	10.000	n/a	0.250	3.973	1.030	Unassigned

Network All Networks / Highlight Window Center

Reports > Storm Drains / Links > Link Configuration

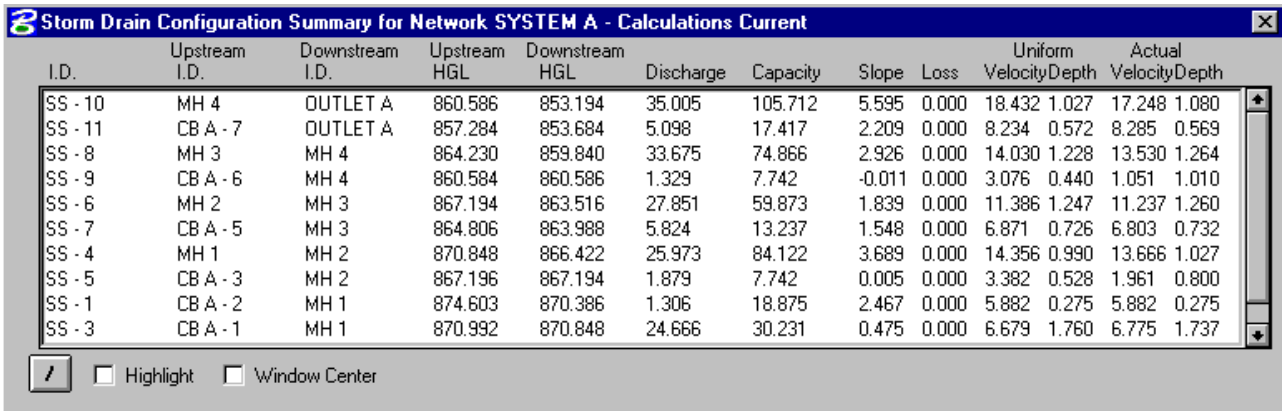


Storm Drain Configuration Summary for Network SYSTEM A - Calculations Current

I.D.	Upstream I.D.	Downstream I.D.	Discharge	Length	Shape	#	Rise	Span	n	Slope	Upstream Invert	Downstream Invert
SS - 10	MH 4	OUTLET A	35.005	132.11	Circul	1	2.500	n/a	0.012	4.891	858.576	852.115
SS - 11	CB A - 7	OUTLET A	5.098	163.00	Circul	1	1.500	n/a	0.012	2.024	856.415	853.115
SS - 8	MH 3	MH 4	33.675	150.00	Circul	1	2.500	n/a	0.012	2.453	862.256	858.576
SS - 9	CB A - 6	MH 4	1.329	19.486	Circul	1	1.500	n/a	0.012	0.400	859.654	859.576
SS - 6	MH 2	MH 3	27.851	200.00	Circul	1	2.500	n/a	0.012	1.569	865.394	862.256
SS - 7	CB A - 5	MH 3	5.824	52.826	Circul	1	1.500	n/a	0.012	1.169	863.874	863.256
SS - 4	MH 1	MH 2	25.973	120.00	Circul	1	2.500	n/a	0.012	3.097	869.111	865.394
SS - 5	CB A - 3	MH 2	1.879	45.560	Circul	1	1.500	n/a	0.012	0.400	866.577	866.394
SS - 1	CB A - 2	MH 1	1.306	170.95	Circul	1	1.500	n/a	0.012	2.377	874.176	870.111
SS - 3	CB A - 1	MH 1	24.666	30.251	Circul	1	2.500	n/a	0.012	0.400	869.232	869.111

/ Highlight Window Center

Reports > Storm Drains / Links > Link Hydraulic Computations

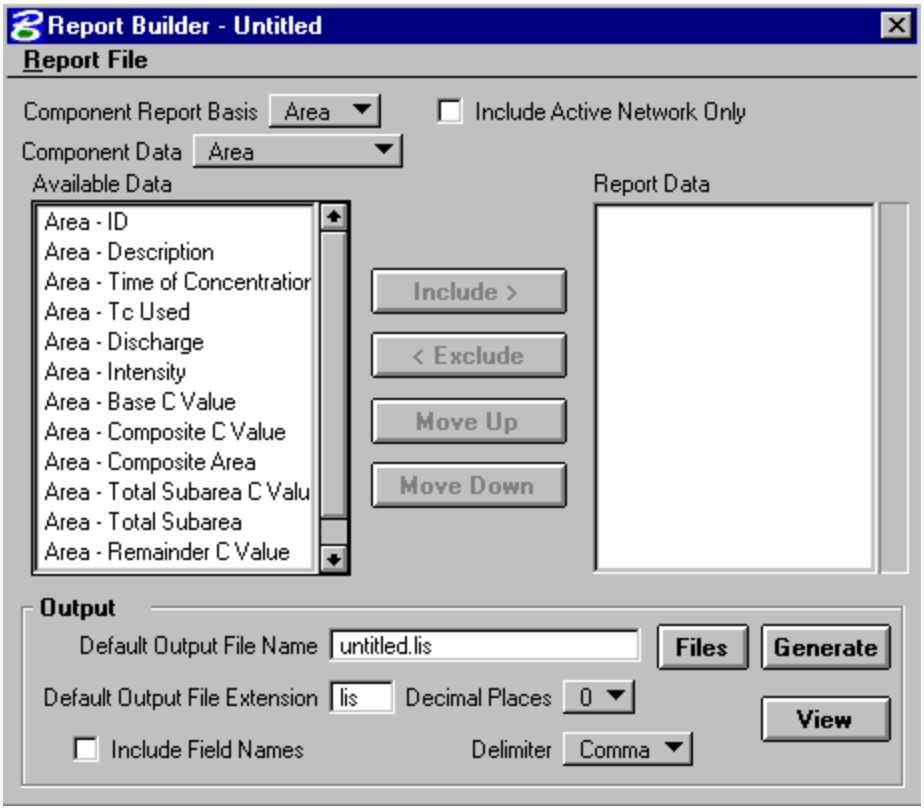


Storm Drain Configuration Summary for Network SYSTEM A - Calculations Current

I.D.	Upstream I.D.	Downstream I.D.	Upstream HGL	Downstream HGL	Discharge	Capacity	Slope	Loss	Uniform Velocity	Uniform Depth	Actual Velocity	Actual Depth
SS - 10	MH 4	OUTLET A	860.586	853.194	35.005	105.712	5.595	0.000	18.432	1.027	17.248	1.080
SS - 11	CB A - 7	OUTLET A	857.284	853.684	5.098	17.417	2.209	0.000	8.234	0.572	8.285	0.569
SS - 8	MH 3	MH 4	864.230	859.840	33.675	74.866	2.926	0.000	14.030	1.228	13.530	1.264
SS - 9	CB A - 6	MH 4	860.584	860.586	1.329	7.742	-0.011	0.000	3.076	0.440	1.051	1.010
SS - 6	MH 2	MH 3	867.194	863.516	27.851	59.873	1.839	0.000	11.386	1.247	11.237	1.260
SS - 7	CB A - 5	MH 3	864.806	863.988	5.824	13.237	1.548	0.000	6.871	0.726	6.803	0.732
SS - 4	MH 1	MH 2	870.848	866.422	25.973	84.122	3.689	0.000	14.356	0.990	13.666	1.027
SS - 5	CB A - 3	MH 2	867.196	867.194	1.879	7.742	0.005	0.000	3.382	0.528	1.961	0.800
SS - 1	CB A - 2	MH 1	874.603	870.386	1.306	18.875	2.467	0.000	5.882	0.275	5.882	0.275
SS - 3	CB A - 1	MH 1	870.992	870.848	24.666	30.231	0.475	0.000	6.679	1.760	6.775	1.737

Highlight Window Center

The Report Builder tool is used to build reports on drainage component features such as Area, Inlet, Node and Link. A particular report format can then be saved for subsequent reuse. The Report Builder dialog box is depicted below.

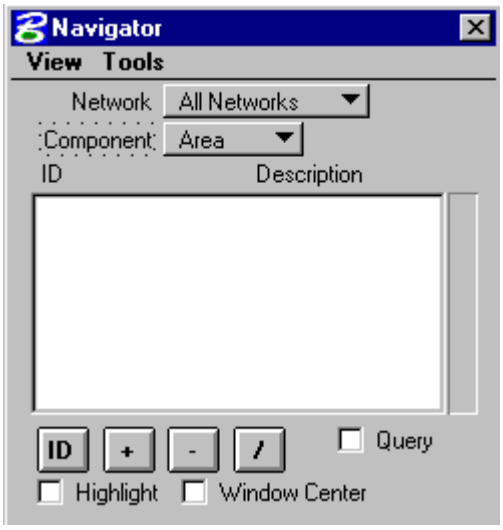


The Report > Generate command can be used to access a custom format saved within the Report Builder. Additional reports can then be generated using the same layout.

TOOLS MENU

NAVIGATOR MENU

The Navigator tool provides a means to identify, add, edit, delete, and query the components of the current project. The Navigator also provides graphical maneuvering tools to move through the design file and view the components with highlighting and window centering functions. When the navigator tool is accessed, a dialog box appears, with the following features:



Network: Either all the networks are selected or only the active network is selected.

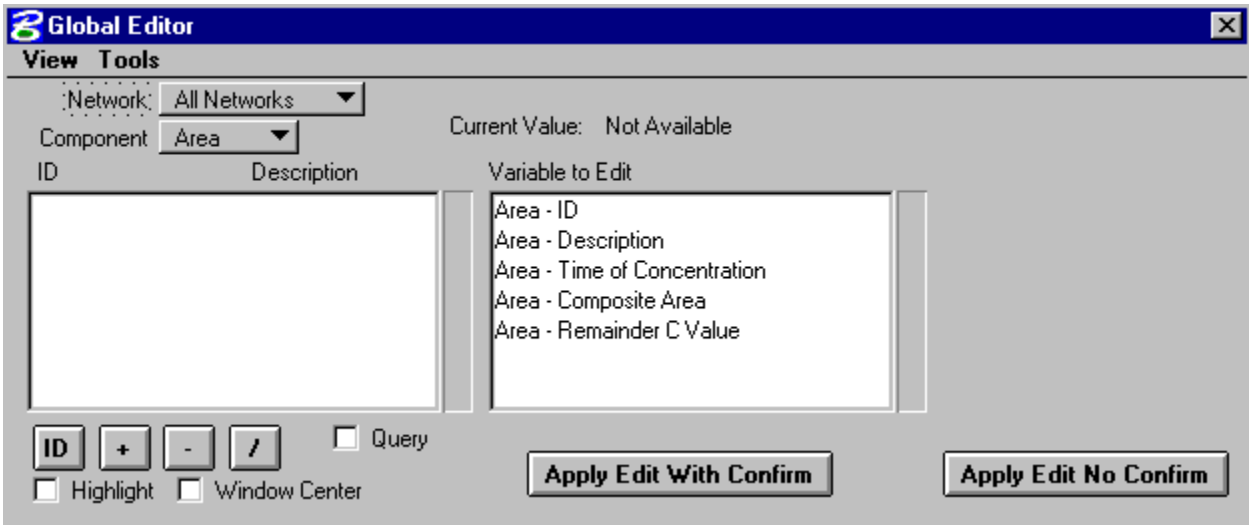
Component: A list box allows one to choose the various components supported by GEOPAK Drainage.

ID/Description List Box: Lists all defined elements of the specified component and their respective descriptions.

Add/Delete/Edit Buttons: These buttons are used to add, delete or modify elements of a specified type and ID.

Query: This check box activates the query tool that searches for components either by a value or a constraint.

The View > Global Editor option in the Navigator tool brings up the global editor dialog box. The Global Editor is used to edit the input variables and parameters of the system under consideration. Numeric variables are edited with the New Value field, and alphanumeric variables use the overwriting text or find and replace text fields.



When the Tools > Update Graphics is selected in Navigator or Global Editor, the graphic representation of the selected components will be updated.

DRAINAGE TOOLS

Exercise of this option brings up a tools palette that allows one to analyze and evaluate drainage patterns. The tools that can be employed are:

CREATE REFINED WATERSHED: This tool is used to refine a Triangulated Irregular Network (TIN) file for use with other Drainage tools. This tool processes the entire network, evaluating ridges, sumps and low points, and determines more precise flow boundaries within the triangles. Selection of this icon invokes a dialog box that prompts for the original TIN file name and the watershed TIN File name which is the refined file to be created.

DELINEATE TIN WATERSHEDS: The Delineate TIN Watershed tool displays the watershed boundaries that exist within a Digital Terrain Model. A watershed is defined by either a low point within the TIN or a low edge point along the TIN hull. MicroStation shapes may be created for each watershed contained in the TIN.

DELINEATE WATERSHED: This tool further delineates watersheds at any location within the network. A boundary element representing the pour point of the watershed is indicated, and the contributing watershed area is computed and delineated. This tool also requires that the overall watershed boundary containing the pour point be selected. This

facilitates subdividing watersheds. The resulting watersheds can then remove previously defined upstream watersheds to depict conditions as would occur if structures upstream are intercepting portions of the contributing watershed.

MAXIMUM DESCENT PATH FROM TRIANGLES: This tool evaluates the flow paths contained in the network. This tool performs a downstream trace from the centroid of each triangle contained in the region specified. The results help provide visualization of the flow patterns within the TIN.

DELINEATE SUMP LINES: A sump line is a triangular edge where the flow on each side of the edge is towards the edge, e.g., as in streams and ditches. This tool indicates the sump lines within a region of the network.

DELINEATE RIDGE LINES: A ridge line is defined as a triangular edge where the flow on each side of the edge is away from the edge. This tool indicates the ridge lines within a region of the network.

DELINEATE LOW POINTS: The Delineate Low Points Tool locates all the low points within a region of a TIN.

DELINEATE HIGH POINTS: The Delineate High Points Tool locates all the high points within a region of a TIN.

DOWNSTREAM TRACE: The Downstream Trace Tool delineates the flow path downstream from a given point in the TIN. The indicated path follows the steepest descent from the point through the TIN ,terminating at a low point or the edge of the TIN.

UPSTREAM TRACE: The Upstream Trace Tool delineates the flow path upstream from a given point in the TIN. The indicated path follows the steepest ascent from the point through the TIN, terminating at a high point or the edge of the TIN.

DELINEATE FLOW DIRECTION: The Delineate Flow Directions Tool indicates the direction of flow within the triangles for a given region of the TIN.

Chapter 2:

FEATURES OF STORMWORKS

FILE MENU

The **File > Open Project** command loads all the information associated with an existing project.

The **File > Save As** command creates a new project and specifies which data files to store there. A project is comprised of surfaces, coordinate geometry projects, a general preference file, and a geometry annotation preference file.

SURFACE COMMANDS

The **File > Surface** commands create, open, and save surface files. Surface files can contain triangle and point information that define a Digital Terrain Model (DTM). A DTM is a numerical representation of a ground surface that is made up of x, y, z coordinates.

The **File > Surface > New** command creates a new, independent surface, which becomes the active surface. The active surface is the surface that receives the input. When this command is used, the software creates a slot in memory where surface data is stored. After this slot in memory is created, point data can be loaded into the surface. There is no

limit to the number of points or triangles that may be loaded into individual surfaces. An unlimited number of independent terrain models can be used simultaneously.

The **File > Surface > Open** command is the primary means of importing existing point and triangle information into a surface. This command can be used after previously creating a surface, loading terrain model data, and then saving the surface.

The **File > Surface > Save** command saves point and triangle information from the active surface to a digital terrain model (DTM) or topological triangulated network (TTN) file.
Note that this command does not prompt before overwriting an existing surface.

The **File > Surface > Save As** command saves point and triangle information from an existing surface to a digital terrain model (DTM) or topological triangulated network (TTN) file.

The Geometry command on the File pull-down menu allows one to create, load, and save geometry project files. These files contain geometric information defining points, alignments, super elevation lists, and their parameters. A geometry project is similar to a database in that it stores and organizes information about several geometry projects at a time. Note that only one geometry project can receive input at a given time.

The **File > Geometry > New Geometry** command allocates memory for a new geometry project. A geometry project needs to be created before working on the design.

It is worthwhile to note here that *Geometry project files are interchangeable between StormWorks, InRoads, InRail, and all subset products such as InXpress, CogoWorks, SiteWorks, and TrackWorks.*

The **File > Geometry > Open** command loads an existing geometry project file into memory.

IMPORT COMMANDS

StormWorks provides various commands to import data. These commands are used to:

(a) Transform MicroStation graphic elements into geometry data

The **File > Import > Geometry From Graphics** command transforms MicroStation graphic elements into geometry data.

(b) Load ASCII files that contains alignment data

The **File > Import > ASCII Geometry** command loads a horizontal or vertical alignment or COGO points by loading an ASCII file that contains the alignment information. One can load horizontal alignments containing any combination of the following elements: tangents, circular curves, and spiral curves. *Note that this command does not support station equations, vertical clothoids or non-linear spirals.*

(c) Load ASCII surface data

The **File > Import > ASCII Surface** command loads digital terrain model data stored in an ASCII text file into a surface. Importing ASCII data allows one to define the coordinates of points of a surface using data collected in the field.

(d) Load digitized model and MicroDrainage data.

The **File > Import > Surface From Graphics** command loads terrain model data digitized in a MicroStation design file into an existing surface. Terrain model data can include digitized contours and graphic elements representing the location of random, breakline, contour, interior boundary, or exterior boundary points. One can select elements from a reference file as input to this command.

The **File > Import > ASCII Model** command reads existing ASCII data files to create a drainage network. This file contains the minimum amount of data necessary to load the network and to design or model it.

The **File > Import > MicroDrainage.SWS** command loads an existing MicroDrainage network file. These files contain network structures based on calculations performed in the MicroDrainage product.

PREFERENCE COMMANDS

For any given command, multiple sets of parameter configurations can be saved to a preference file. The individual sets of saved parameters, known as preference sets, can be recalled from the preference file. This provides for reuse wherein different drainage projects can make use of the same set of preference files.

The **File > Open Preference > General** command opens a preference file, allowing one to gain access to a different set of general parameters. General preference files define

preferences for everything except coordinate geometry elements (alignments and geometry points) and horizontal element annotation.

The **File > Open Preference > Geometry Annotation** command opens a different preference file, allowing one to access a different set of geometry parameters. Geometry preference files define preferences for coordinate geometry elements only.

Using the **File > Save As > General** command saves a different preference file to gain access to a different set of parameters at another time. Many commands contain numerous parameters governing its processing. These parameters can be stored in a file known as a preference file.

The **File > Save As > Geometry Annotation** command saves a user-defined preference file, facilitating recall of a set of geometry parameters.

PARAMETER COMMAND

The parameter file commands located on the File pull-down menu allow one to open and save default parameters for placing StormWorks structures.

The **File > Active Parameters > Open** command loads previously saved design project parameters and default criteria into the active design from a specified ASCII file.

The **File > Active Parameters > Save** command saves the active StormWorks parameters into an ASCII parameter file. These stored parameters are then recovered automatically when the **File > Active Parameters > Open** command is chosen.

The **File > Active Parameters > Save As** command saves an existing parameter file to a different name or a different directory. When the file is saved, it is stored on the hard-disk drive, and can be opened at any point during a design session.

STYLE COMMANDS

The Style commands help in annotating profiles and the network data. They are used to set up line styles, font types, colors, and text sizes.

The **File > Style > Open** command opens a file that defines the appearance of StormWorks elements. The level, color, line style, and line weight of graphics of StormWorks elements such as pipes and nodes can be defined. The **File > Style > Save** command saves the current style file.

ANNOTATION SETTINGS COMMANDS

The Annotation Settings commands located on the File pull-down menu allow one to define annotation parameters. These files are used to set up annotation parameters, such as font, text size, color, weight, and label placement, when annotating both networks and profiles.

QUIT COMMAND

To exit StormWorks and keep MicroStation active, the File > Quit command is used. When this command is chosen, a check is made to verify that the project data has been saved. The user is prompted to save the project in case this command is executed after changes are made to the project.

SETTINGS MENU

The Settings commands provide a framework for all other StormWorks commands. The Settings command can be used in the following ways**:**

- The write lock can be turned on and off.
- Preferences can be set for units and formats.
- Parameters can be set for placing and displaying drainage structures in designs.
- Parameters for computing area drainage and runoff can be set.
- Parameters for annotating structures and profiles, select flow and time to inlet equations for calculations can be set.
- Parameters for MicroDrainage calculations can be set.

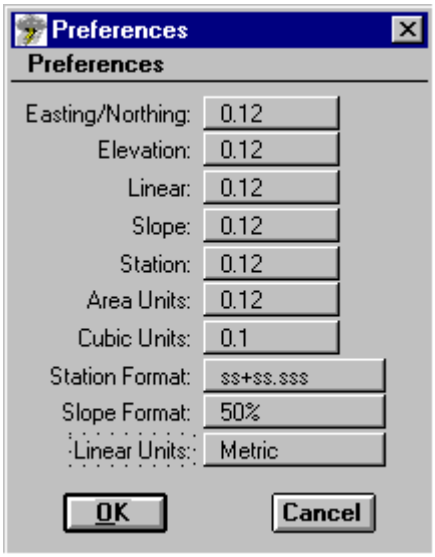
LOCKS COMMANDS

The **Settings > Locks** command allows access to the Write, Snap, Station, Report, and Dialog locks.

The **Settings > Locks > Dialog** command opens a dialog box that facilitates review and change of locks quickly and dynamically.

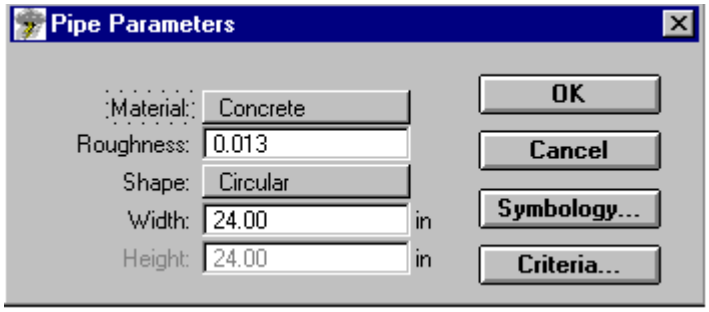
PREFERENCES COMMAND

The **Settings > Preferences** command opens the Preferences dialog box. The Preference dialog box is used to define preferences for precision, format, and units. A set of default preferences for these parameters is displayed on the dialog box.



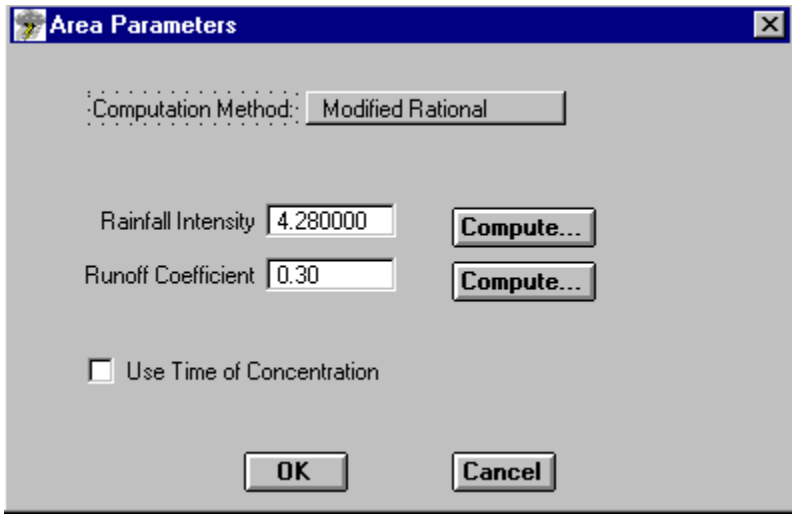
PIPES COMMAND

The **Settings > Pipes** command opens the Pipe Parameters dialog box, which lets one set parameters that control the layout and design characteristics of a pipe. These parameters are used when placing pipes in the design. Similar commands are used for the layout and design of inlets, manholes, channels, culverts, and pumps.



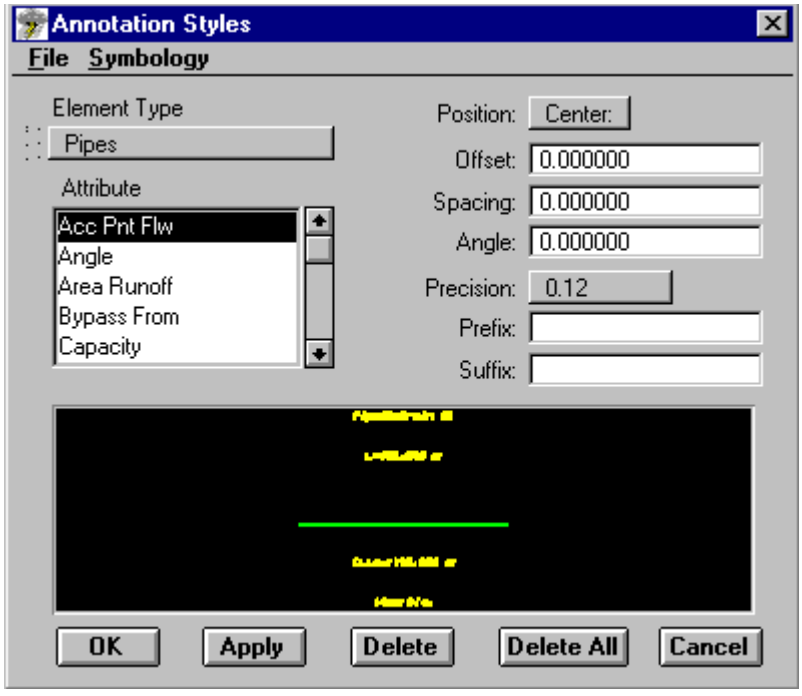
AREAS COMMAND Palettes

The **Settings > Areas** command opens the Area Parameters dialog box, which is used to set parameters required for computing drainage area and peak runoff (flow rate). The computation method is selected among *the Modified Rational method or SCS unit hydrograph* method, and the required parameters are entered, which vary depending on the method. The **Flow > (Compute Flow**) command can then be used to compute the area and runoff of a selected shape (polygon).



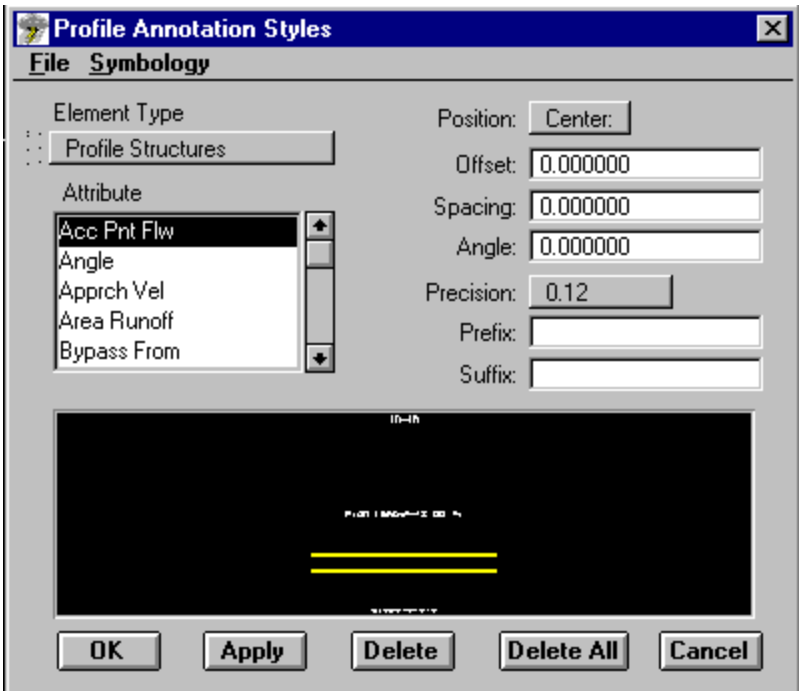
ANNOTATION COMMAND

The **Settings > Annotation** command opens the Annotation Styles dialog box, which is used to define parameter values for textual labels for structural design elements, such as pipes, manholes, inlets, channels, and so on. A label can be assigned to each attribute of a given element type (structure). Once the element type and attribute are selected text symbology, positioning parameters and the label name prefix and suffix values can be defined. Text symbology includes items such as color, font, height, etc. Positioning parameters are position, offset, spacing and angle. The parameter values can be saved to an annotation (.ann) file that can be opened later to retrieve a particular set of values.



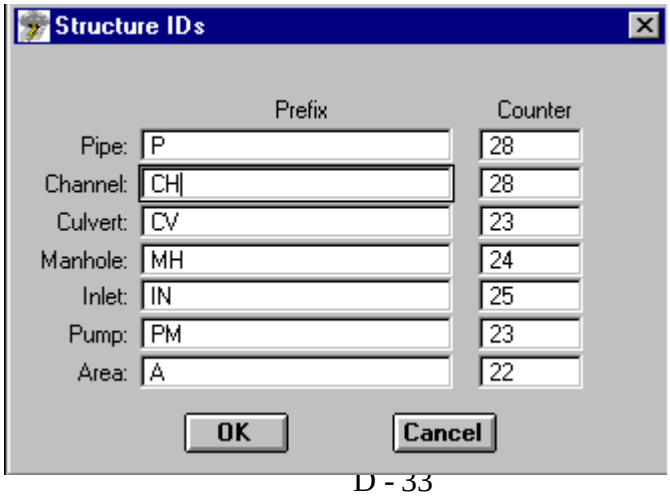
PROFILE ANNOTATION COMMAND

The **Settings > Profile Annotation** command opens the Profile Annotation Styles dialog box, which is employed to define parameter values for textual labels for profile structures and junctions. Attributes can be assigned similar to the annotation command.



STRUCTURE IDs COMMAND

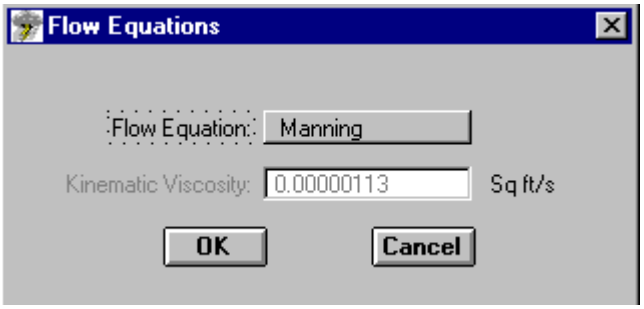
The **Settings > Structure IDs** command opens the Structure IDs dialog box.



The Dialog box is utilized to review and define identification names (IDs) for each structure. The default structure IDs consist of a one or two character prefix that describes the structure type, followed by a counter number that is automatically assigned (in sequential order) to each structure as it is placed in the design. The IDs can also be customized to meet the user's needs.

FLOW EQUATIONS COMMAND

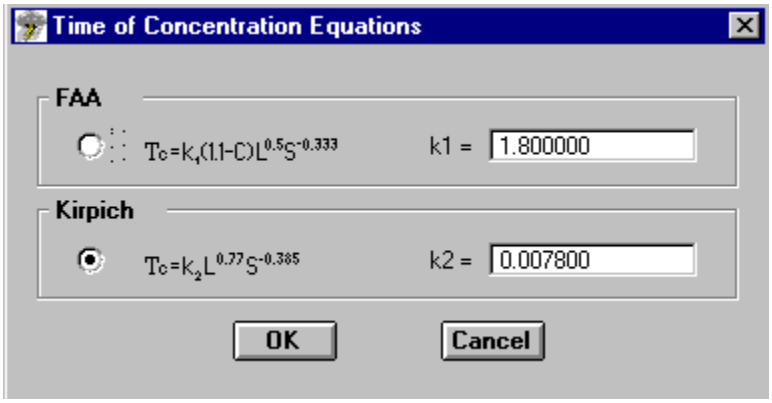
The **Settings > Flow Equations** command activates the Flow Equations dialog box, prompting the user to select the equation to use for computing discharge flow rate. The equations that can be selected are either the Manning or Darcy/Colebrook equations.



TIME EQUATIONS COMMAND

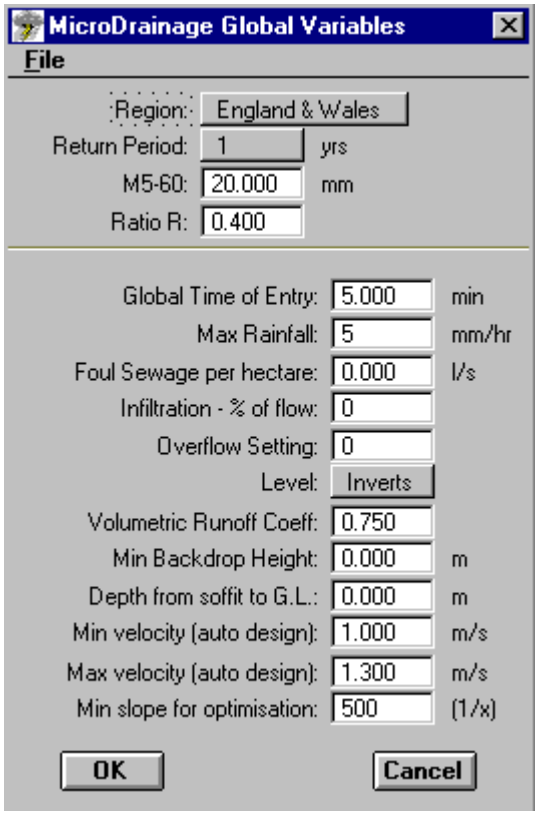
The **Settings > Time Equations** command opens the Time of Concentration Equations dialog box, prompting the user to select the equation to use for computing the time of flow to an inlet. Two equations are provided: the FAA flow time equation or the Kirpich flow time equation. These equations are used in conjunction with the **Palettes > Flow > (Inlet**

Time) command. Note that the runoff coefficient of the drainage area must be computed using the **Settings > Areas** command, prior to using the time equations.



MICRO DRAINAGE GLOBALS COMMAND

The **Settings > MicroDrainage** globals command opens the MicroDrainage Global variables dialog box. This dialog box is used to define and save parameter values for analyzing and designing a drainage network using standard MicroDrainage modules. This


MicroDrainage Global Variables
File
Region: England & Wales
Return Period: 1 yrs
M5-60: 20.000 mm
Ratio R: 0.400
Global Time of Entry: 5.000 min
Max Rainfall: 5 mm/hr
Foul Sewage per hectare: 0.000 l/s
Infiltration - % of flow: 0
Overflow Setting: 0
Level: Inverts
Volumetric Runoff Coeff: 0.750
Min Backdrop Height: 0.000 m
Depth from soffit to G.L.: 0.000 m
Min velocity (auto design): 1.000 m/s
Max velocity (auto design): 1.300 m/s
Min slope for optimisation: 500 (1/x)
OK
Cancel

is a two way process. The values, which are stored in a standard stormwater file, are exported into MicroDrainage. Upon completion of MicroDrainage analyses and design, the stormwater file is imported back to StormWorks.

VIEW MENU

The View commands provide the tools necessary to display data stored in digital terrain-model surfaces and the active coordinate geometry project. These views are for the active surface and geometry. Many of these commands can also be accessed using StormWorks Palettes menus.

FIT SURFACE COMMAND

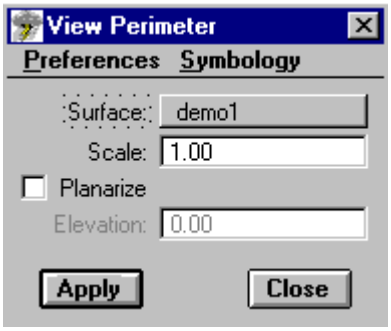
The **View > Fit > Surface** command fits a graphic view using graphics associated with the active surface. This command is helpful if the graphics associated only with a single surface is to be viewed. This command does not fit coordinate geometry graphics into the view. In addition, MicroStation graphics in the design file that are a different elevation from the surface graphics may not appear. Only one surface can be fit at a time.

FIT ALIGNMENT COMMAND

The **View > Fit > Alignment** command reorients and resizes the selected view about the extents of an alignment. Only one alignment can be fit at a time.

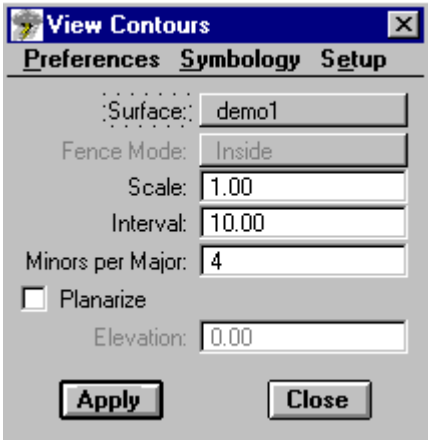
VIEW PERIMETER COMMAND

The **View > Perimeter** command displays the outermost edge of a triangulated digital terrain-model. This command is helpful for visualizing the boundaries of the triangulated model and for fitting the surface to the viewing screen so that the results of subsequent commands can be seen.



VIEW CONTOUR COMMAND

The **View > Contours** command generates and displays elevation contours for the active surface. This command automatically generates major contours, minor contours, and optionally, contour labels in the design file.



DEPRESSION CONTOURS COMMAND

The **View > Depression Contours** command displays depression contours for the active surface. Depression contours incrementally descend the surface and are represented with tick marks.

LABEL CONTOURS COMMANDS

The **View > Label Contours** command interactively labels contours in a digital terrain model. This command is helpful if one wants to specify where contour labels will appear.

VIEW TRIANGLES COMMAND

The **View > Triangles** command displays all or a portion of the triangles contained within the active surface. The triangles are created when the model is triangulated. This command is useful after triangulating the data so that the location of surface points can be seen. This command can be used to evaluate a model and pinpoint areas that either have too many points or too few points.

VIEW DTM POINT ELEVATIONS COMMAND

The **View > DTM Point Elevations** command finds the elevation (z coordinate) of any point in the surface. This command automatically annotates an area of the surface, or is used to manually identify individual locations on the surface at which to display elevations.

VIEW SLOPE VECTORS

The **View > Slope Vectors** command displays vector lines and arrowheads to show the downhill direction and degrees of slopes at various points throughout a surface. The length of each vector is proportional to the slope of the model; longer vectors indicate steeper slopes.

TWO POINT SLOPE COMMAND

The **View > Two Point Slope** command is used to display the slope between two points. The slope is represented as a linestring based on the active MicroStation text and symbology settings.

VIEW DRAPE SURFACE

The **View > Drape Surface** command projects existing MicroStation elements to the elevation of the active surface and then redisplays those elements. This command drapes lines, linestrings, curves, shapes, arcs, ellipses, and complex strings or shape combinations of these elements. Text, nodes, and cells can also be draped.

VIEW COLOR CODED COMMAND

The **View > Color-Coded** commands display a surface in various colors based on the elevation of the points in the surface. The surface may be color coded by elevation, slope, or aspect. Each of these commands generates MicroStation shape elements and can produce a complete ASCII report showing area quantities for each elevation, slope, or aspect range. Additionally, when used in conjunction with any of the available

MicroStation rendering commands, the Color-Coded commands yield accurate depictions of a site's elevation, slopes, or aspects.

VIEW ACTIVE ALIGNMENT

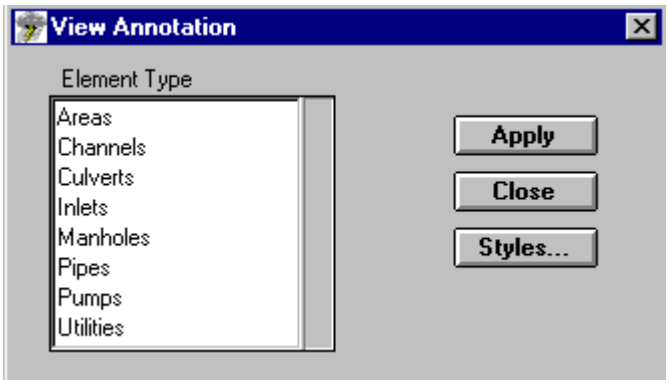
The **View > Active Alignment** command displays the active horizontal alignment in the design file.

VIEW STATIONING COMMAND

The **View > Stationing** command activates the View Stationing dialog box, which controls the display of station annotation text along the active horizontal or vertical alignment. Stations can be placed along a horizontal or vertical alignment at a specified interval.

ANNOTATE COMMAND

The **View > Annotate** command displays labels for attributes, such as height, length, ID, invert elevation, contributing area, discharge, and material type for different StormWorks drainage structures.



ANNOTATE PROFILE COMMAND

The **View > Annotate Profile** command displays labels for attributes such as ID, slope, length, and material type of drainage structures and junctions in a selected profile.

CONTRIBUTING NETWORK COMMAND

The **View > Contributing Network** command allows one to view all of the elements that contribute to a network.

PALETTES COMMANDS

The Palettes commands are used to perform the majority of the graphical placement and manipulation functions of StormWorks. These commands display palettes from which a wide variety of tools can be chosen to view surface and geometry information. One can also place, design, and manipulate drainage structures, compute drainage information, create and edit profiles and HEC-2/WSPRO sections, import HEC-2 water surfaces, and display hydrologic information.

MAIN COMMANDS

The Main palette contains icons that represent all of the StormWorks commands available on the Palettes pull-down menu. This provides a shortcut to frequently used commands--- The Main palette can be displayed rather than going through the Palettes menu. Beneath each icon on the Main palette lies the corresponding group of command icons represented as tear-off palettes. Options offered are View Surface palette, View Geometry palette,

Layout palette, Flow palette, Design palette, Manipulate palette, Profiles palette, and HEC/WSPRO palette.

VIEW SURFACE

The **Palettes > View Surface** commands allows one to display the various elements of a surface. Elements that can be displayed are perimeter, contours, depression contours, label contours, triangles, DTM point elevations, slope vectors, two point slope, drape surface, color coded elevations, color coded slopes, color coded aspects, and color coded triangles.

VIEW GEOMETRY

The **Palettes > View Geometry** commands control the display and annotation of horizontal and vertical alignments, horizontal and vertical stations, 3-D alignments, points, and profiles. This command affords control over the alignments and elements to be displayed as well as their annotation. Sub-commands are viewing the active alignment and stationing and fitting the alignment.

LAYOUT COMMANDS

The **Palettes > Layout** commands allow one to interactively place drainage structures, such as pipes, inlets, channels, manholes, and pumps. These commands are usually used after a new project design file is set up. This includes defining the active project parameters, the minimum and maximum design criteria, and the display characteristics of

the structures. After loading and defining the active surface, the Layout commands are used to place the structures needed to design the storm drainage system.

FLOW COMMANDS

The **Palettes > Flow** commands allow one to define percentages of land cover and compute peak runoff (flow rate) resulting from a storm. These commands are used to define the parameters for calculating rainfall intensity and to determine the type of equation best suited for computing inlet time of flow. Other features of this command allow one to get flow into the system, establish the discharge parameters, compute drainage areas and peak runoffs, attach computed drainage areas to a specified structure, highlight the attached areas, and highlight inlets not attached to drainage areas. Additionally one can attach bypass inlets, compute time of flow, highlight inlets with or without routed bypass, and detach the computed area flow or injected flow from a specific structure.

DESIGN COMMANDS

The **Palettes > Design** commands is used to design a new storm drainage system. One can design an individual structure or downstream structures that are affected by discharge from upstream structures. Additionally this command can be used to analyze pipe design characteristics, compute pipe sizes, and analyze channel design characteristics.

Note that when designing structures, StormWorks displays a warning if design criteria are exceeded. If the report lock is toggled on, a dialog box appears displaying the warning messages.

MANIPULATE COMMANDS

The **Palettes > Manipulate** commands is used to fine-tune the drainage network and to manipulate and modify the information stored on the StormWorks structures. These commands are used to adjust the layout to satisfy the minimum cover criteria, to renumber network connections, and to review/edit the structure parameter values.

PROFILES COMMAND

The **Palettes > Profile** commands allow one to work with profiles of the drainage system. This command can be used to create a profile of a drainage system, update existing profiles, set parameters for and display profile annotation, display graphics where linear features intersect a profile, check pipe cover between profile structures, insert manholes and inlets into a profile, and move profile structures.

HEC/WESPRO COMMANDS

The **Palettes > HEC/WSPRO** commands is used to create HEC-2 and WSPRO channel sections, to import HEC-2 and WSPRO water surfaces, and to plot water surface profiles.

HYDROLOGY COMMANDS

The **Palettes > Hydrology** commands allow one to display features, such as contours, ridges, flat areas and valleys that affect water flow on the surface of the DTM. One can also display flow paths and tributaries and compute the volume of water in retention ponds.

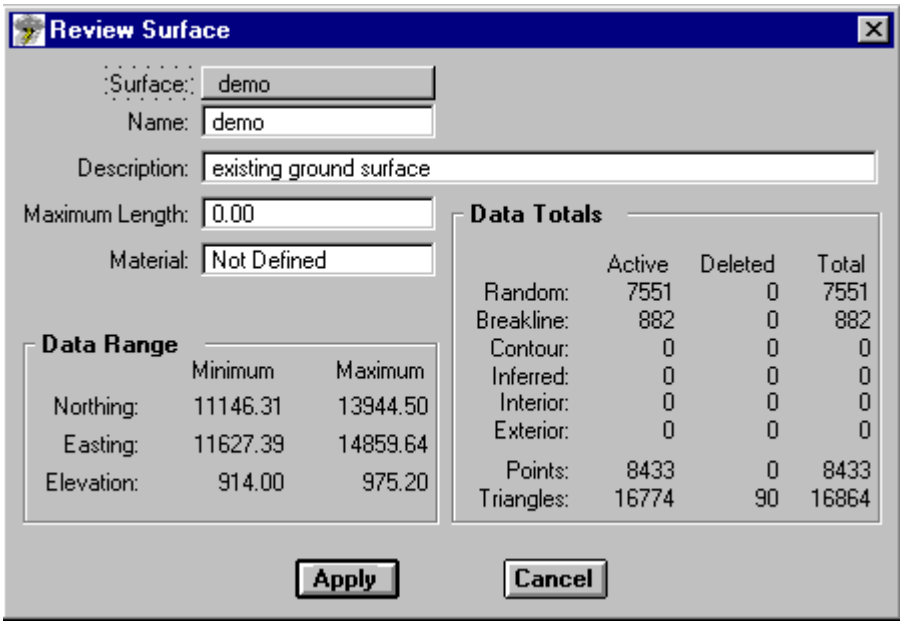
UTILITIES COMMANDS

The Utilities commands allow one to perform general design file tasks that are appropriate throughout the design cycle, such as manipulating and reviewing the active geometry.

The **Utilities > Active Surface** command allows one to designate any existing surface as the active surface. The surface should have been created or opened before using this command.

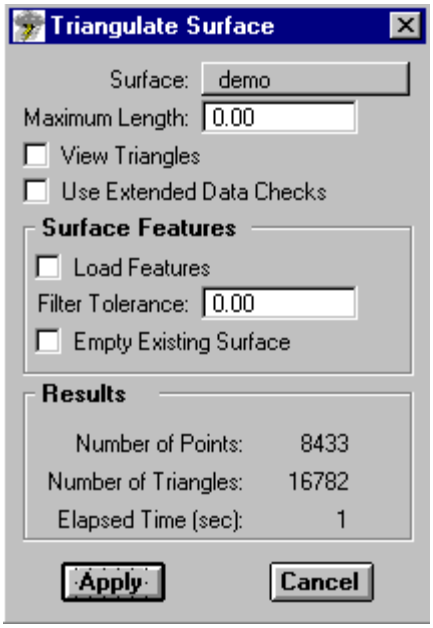
The **Utilities > Delete Surface** completely deletes a surface and removes the point and triangle data associated with the surface. This command is used when incorrect data is loaded onto the surface.

The **Utilities > Review Surface** command reviews information associated with any surface. This command can be used to review the number of points and triangles in a surface. Modifications can be made to the name, description and maximum triangle side length parameters associated with any surface.



The **Utilities > Channel To Surface** command merges the designed channel surface into an existing surface. The resulting surface is a fully triangulated new surface containing the original surface data plus the channel.

The **Utilities > Triangulate Surface** command is used to form or re-form the 3-D faceted surface that defines a digital terrain model. Small, triangular plains are created by the exercise of this command.



The **Utilities > Active Geometry** command changes the active geometry project, alignments and superelevation. This command can also be used to display and update information. The geometry types that can be activated are Geometry Project, horizontal alignment, vertical alignment and superelevation.

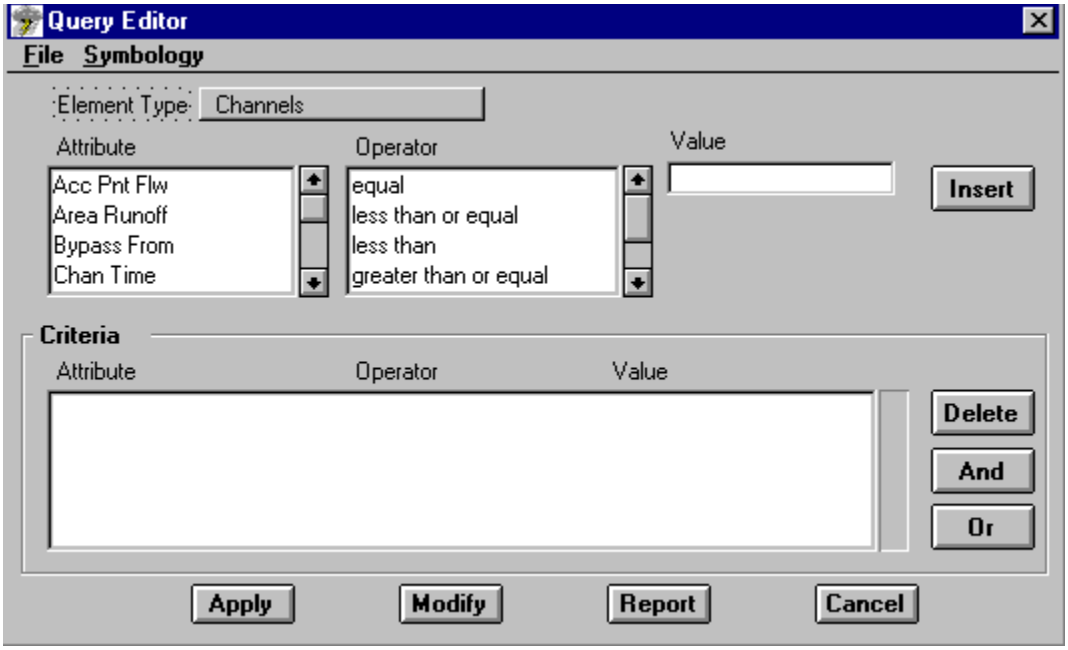
The **Utilities > Delete Geometry** command removes from internal memory a geometry project, a horizontal alignment or a vertical alignment and all its children, or a superelevation. Once such a geometry has been deleted, it is removed from memory and is not recoverable.

The **Utilities > Review Geometry** command displays information about the geometry of the active horizontal alignment.

The **Utilities > Tracking > General** command dynamically displays the northing and easting coordinates of the cursor location as well as the surface elevation, slope and aspect.

The **Utilities > Tracking > Geometry** command tracks the cursor along the active horizontal alignment and displays the station, offset, and elevation of the alignment in the MicroStation Command Window.

The **Utilities > Queries and reports** command allows one to locate, modify and generate reports on drainage elements using queries. The Query dialog box provides options that define the query criteria. Once this command is executed, StormWorks displays the graphics that represent all the network elements meeting the query criteria.



The **Utilities > Maintenance Information** command provides for maintaining a service record for the structures in the network. The parameters employed are the identification number of the selected structure, date the structure was installed, and the date the structure was last serviced.

The **Utilities > Promote Micro Drainage** command allows for naming the structures in a drainage network compatible with the MicroStation naming system. The network can then be saved to a .sws file for further analysis.

The **Utilities > DraftWorks** command activates the DraftWorks product.

APPENDIX E

SOURCE CODE FOR THE CULVERT PROTOTYPE